In this user manual we have tried to describe the matters concerning the operation of this CNC system to the greatest extent. However, it is impossible to give particular descriptions for all unnecessary or unallowable operations due to length limitation and products application conditions; Therefore, the items not presented herein should be regarded as “impossible” or “unallowable”.

Copyright is reserved to GSK CNC Equipment Co., Ltd. It is illegal for any organization or individual to publish or reprint this manual. GSK CNC Equipment Co., Ltd. reserves the right to ascertain their legal liability.

PREFACE

Dear Users,

We are honored by your purchase of the products made by GSK CNC Equipment Co., Ltd.

The manual describes the programming, operation, installation and connection of this GSK96 multi-function position control system in detail.

To ensure safe and effective running, please read this manual carefully before installation and operation.

Notes before operation:

- Connect the ESP button of the system firmly and correctly. As the system uses the normal-closed contact, an ESP alarm will occur upon power on if the button is poorly connected or connected as the normal-open contact, and the system cannot work properly (it does not belong to system fault).
- Set the program reference point of the system according to the actual mounting position of the tool. Using program reference point function before setting the point may cause unexpected accidents.

Warning

Accident may occur by improper connection and operation！This system can only be operated by authorized and qualified personnel.

Special caution:

The power supply fixed on/in the cabinet is exclusively used for the CNC system made by GSK.

It can't be applied to other purposes, or else it may cause serious danger!

ANNOUNCEMENT!

- **This manual describes various possibilities as much as possible. However, operations allowable or unallowable cannot be explained one by one due to so many possibilities that may involve with, so the contents not specially stated in this manual shall be regarded as unallowable.**

WARNING!

- **Please read this manual and a manual from machine tool builder carefully before installation, programming and operation, and strictly observe the requirements. Otherwise, products and machine may be damaged, workpiece be scrapped or the user be injured.**

CAUTION!

- **Functions, technical indexes (such as precision and speed) described in this user manual are only for this system. Actual function configuration and technical performance of a machine tool with this CNC system are determined by machine tool builder's design, so functions and technical indexes are subject to the user manual from machine tool builder.**
- **Though this system adopts standard operation panel, the functions of the keys on the panel are defined by PLC program (ladder diagram). It should be noted that the keys functions described herein are for the standard PLC program (ladder diagram).**
- **For functions and effects of keys on control panel, please refer to the user manual from machine tool builder.**

This manual is subject to change without further notice.

Suggestions for Safety

The user must carefully read the suggestions for the system before installing and operating the system.

The user must follow the suggestions of the system to ensure that the person is not hurt and the equipments are not damaged.

The user must follow the related suggestions for safety described in the user manual, and must not operate it until the manual is read completely.

The user must follow the suggestions of safety described in the user manual from the machine manufacture.

The user can operate the machine or compile the program to control the machine after completely reading the manual and the one from the machine manufacturer.

I. Graphic symbol

Caution Operation against the instructions may cause the operator serious injuries.

Alarm Wrong operation may injure the operator and damage the system.

Warning Improper operation may result in damage to the machine, as well its products.

! Important information

II. Notes

1) Check before acceptance

Caution ● The damaged or defect product must not be used.

2) Delivery and storage

Notice ●Moistureproof measures are needed while the system is delivered and stored. Never climb the packing box, stand on it, or place heavy items on it. Do not put over five packing boxes in piles. Take particular care of the front panel and the display of the system.

3) Installation

Caution ●Protect the system from sunlight and raindrops. The shell of the system is not waterproof.

Notice ●Prevent dust, corrosive air, liquid, conductors and inflammable substances from entering the system.

●Keep the system away from inflammable and explosive substances. Avoid places where there is powerful electromagnetic interference.

●Install the system firmly without vibration.

4) Wiring

Warning ●Only qualified persons can connect the system or check the connection. The connecting wires cannot be damaged. Do not press or open the cover of the system with power on.

Caution ●The voltage and the polarity of connecting plugs must accord with the user manual.

●Wet hands are dangerous to grasp the plug or the switch.

Notice ●The connection must be proper and firm.

●The system must be earthed.

5) Debugging

Caution ●Make sure that the parameters of the system is correct before the system runs.

●No parameter is beyond the setting limit in the manual.

6) Operation

Warning ●Only qualified operators can operate the system.

●Ensure the switch is OFF before connecting the power supply.

Notice ●The operator cannot leave the system to work alone.

●Do not switch on the system until making sure the connection is correct.

●The emergency stop button is able to disconnect all power supplies when the system breaks down. Do not switch on/off the system frequently.

Caution ●Prevent the system from the environmental interference.

7) Troubleshooting

Warning ●Unqualified persons cannot repair the system.

Caution ●After alarms; do not restart the system until the breakdown is fixed.

III. Safety suggestions for programming

1）Setting a coordinate system

Incorrect coordinate system may cause the machine not to work as expected even if the program is correct, which may injure the operator, and damage the machine as well as its tool and workpiece.

2） Rapid traverse (positioning)

When G00 rapid traverse performs the positioning (nonlinear motion to position between its starting point and end point), make sure that the path for the tool is safe before programming. The positioning is to perform the rapid traverse, and when the tool and the workpiece are interfered, the tool, the machine and the workpiece may be damaged, and even the operator injured.

3） Applicability of user manual

The manual introduces in detail all functions of the system, including optional functions and max. controllable ranges, which are subject to change with the machine. Therefore, some functions described in this manual will not work out for other specified machines.If there is any doubt, please read the instruction for the machine.

4） Functions of CNC system and machine

CNC machines depend on CNC systems, but also power voltage cabinets, servo systems, CNC and the operator panels. It is hard to explain all the integrated functions, programming and operation. Do not use integrated instructions not included in the manual until they have been tested successfully.

IV. Notes and Safety Suggestions for Operating Machine

1) Before parts processing

Test the machine without workpiece or tools. Make sure that the machine runs well before it starts to work.

2) Before operating the machine

Check the input data of the system carefully before operating the machine. Incorrect input data may cause the machine to work improperly, and damage the workpiece and the tool, as well as injure the operator.

3) Adaptations between appointed federate and machine operations

Make sure that the input feedrate of the system is suitable for the expected operation. Feedrate has a maximum for each machine, and the amount of the feed rate is subject to change with operation. Choose the maximum according to the instructions of the machine. Improper feedrate leads the machine to work wrongly, and damage the workpiece and the tool, as well as injure the operator.

4) Compensation function

When offset is needed, check the direction and the amount of the compensation. Improper compensation causes the machine to work wrongly, and damage the workpiece and the tool, as well as injure the operator.

5) Manual operation

If the machine is to run in JOG working mode, check the current position of the tool and the workpiece, and correctly specify the moving axis, moving direction and the feedrate. MPG (Hand wheel) control with great override, such as 100%, rotate MPG (used to call electric hand wheel), then the tools and workbench will move fast, which may lead the stop of the MPG. However, the tools and workbench will not stop immediately. The high override of MPG movement may damage the machine and its tool, even injure the operator.

6) MPG returns to the reference point

If the tool is return to the reference point, make sure that the machine has been equipped with the device to detect the reference point; otherwise, the tool cannot reach the reference point, which may damage the machine and its tool, and even injure the operator.

Safety Responsibility

Manufacturer's Responsibility

——Be responsible for the danger which should be eliminated and/or controlled on design and configuration of the provided CNC systems and accessories.

——Be responsible for the safety of the provided CNC systems and accessories.

——Be responsible for the provided information and advice for the users.

User's Responsibility

——Be trained with the safety operation of CNC system and familiar with the safety operation procedures.

——Be responsible for the dangers caused by adding, changing or altering on original CNC systems and the accessories.

——Be responsible for the failure to observe the provisions for operation, adjustment, maintenance, installation and storage in the manual.

All specifications and designs herein are subject to change without further notice.

This manual is reserved by end user.

We are full of heartfelt gratitude for your support by using GSK's products.

Chinese version of all technical documents in Chinese and English languages is regarded as final.

Operation

Introduces operation methods, technical specifications and parameter settings of GSK96multi-function position control system.

Programming

Introduces command codes and program format of GSK96multi-function position control system.

Connection

Introduces installation and connection methods of GSK96multi-function position control system.

Appendix

Introduces supplementary information to installation and connection of GSK96multi-function position control system.

CONTENTS

OPERATION ····· *1*

CHAPTER ONE OVERVIEW ····· **1**

CHAPTER TWO TECHNICAL SPECIFICATIONS ····· **3**

2.1 GSK96 Technical specifications ····· 3

CHAPTER THREE OPERATION PANEL ····· **5**

3.1 LCD Display ····· 5

3.2 LED Status Indicator ····· 5

3.3 Keyboard ····· 5

3.3.1 Character Keys ····· **5**

3.3.2 Working Mode Selection Key ····· **5**

3.3.3 Function Keys ····· **6**

3.3.4 Cycle Start and Cycle Pause (Feed Hold) Key ····· **7**

3.3.5 Manual Axis Control Key ····· **7**

3.3.6 Manual Auxiliary Function Key ····· **7**

3.3.7 Edit Keys ····· **8**

3.3.8 Reset Key ····· **8**

CHAPTER FOUR SYSTEM OPERATION ····· **9**

4.1 System ON/OFF, Initial State, Modal, and Safe Protection ····· 9

4.1.1 Power On ····· **9**

4.1.2 Power Off ····· **10**

4.1.3 System, Program Initial and Modal ····· **10**

4.1.3.1 The Initial State and Modal of the System ····· 10

4.1.3.2 Initial Mode and Modal of Program ····· 10

4.1.4 Safe Protection ····· **11**

4.1.4.1 Hardware Limit Protection ····· 11

4.1.4.2 Software Limit Safe Protection ····· 12

4.1.4.3 ESP Alarm (Emergently Stopping the System) ····· 13

4.1.4.4 Drive Unit Alarm ····· 15

4.1.4.5 Other Alarms ····· 15

4.1.4.6 Switching off Power Supply ····· 15

4.1.4.7 Reset Operation ····· 16

4.2 CNC Working Mode Selection ····· 17

4.3 EDIT Working Mode ····· 17

4.3.1 Part Program Catalog Search ····· **18**

4.3.2 Selection, Creation, Deletion, Rename and Copy a Part Program ····· **18**

4.3.2.1 Selecting and Creating a Part Program ····· 19

4.3.2.2 Delete a Part Program ····· 19

4.3.2.3 Deleting All Part Programs ····· 20

4.3.2.4 Renaming a Part Program ····· 20

4.3.2.5 Copy a Part Program ····· 21

4.3.3 Part Program Communication ····· **21**

4.3.3.1 Sending Part Programs(CNC→USB) ····· 21

4.3.3.2 Receiving Part Programs (USB→CNC) ····· 22

4.3.3.3 TXT Part Program Standard Format in U Disc ····· 22

4.3.4 Part Program Content Input and Edit ····· **23**

4.3.4.1 Inputting Program Content ····· 25

4.3.4.2 Inserting Program Line ····· 26

4.3.4.3 Deleting a Block ····· 26

4.3.4.4 Inserting a Character in a Block ····· 26

4.3.4.5 Deleting a Character in a Block ····· 27

4.3.4.6 Modifying a Block Content ····· 27

4.3.4.7 Program Stored Space ····· 27

4.3.4.8 No. 253 Program Operation ····· 27

4.3.4.9 No. 254 Program Operation ····· 28

4.3.5 hp1 Function ····· **28**

4.3.5.1 Part program Command Help ····· 28

4.3.5.2 Inserting Macro String ····· 29

4.3.5.3 Line Number Sort ····29
4.3.5.4 Replacing Character String ····29
4.3.5.5 Cursor Position ····29
4.3.5.6 MPG Controlling Cursor Moving ····30
4.3.6 Part Program Compiling ····30
4.3.6.1 hp2 Compiling Command ····30
4.3.6.2 Program Compiling Result Analysis ····31
4.3.6.3 Program Compound Check Prompt ····31
4.4 JOG Working Mode ····32
4.4.1 Coordinate Axis Movement ····34
4.4.1.1 JOG Movement ····34
4.4.1.2 Step Movement ····34
4.4.1.3 MPG Control ····35
4.4.1.4 Rapid Traverse Speed Selection ····36
4.4.1.5 Low Speed Feed Speed Selection ····37
4.4.1.6 Inputting Field Moving, Setting Feedrate ····37
4.4.1.7 Drive Unit Enabling Control ····39
4.4.1.8 Coordinate Axis Motion Alarm Prompt ····39
4.4.2 Creating Coordinate System ····40
4.4.2.1 Creating Machine Coordinate System - Machine Zero Return(Machine Reference Point Return) ····40
4.4.2.2 Creating Machine Coordinate System - Without Machine Zero(Without Machine Reference Point) ····41
4.4.2.3 Setting Workpiece Coordinate System ····42
4.4.2.4 Setting Program Reference Point ····43
4.4.2.5 Program Reference Point Return ····44
4.4.2.6 Recovering the Workpiece Coordinate System and Program Reference Point ····44
4.4.3 Spindle Control Function ····44
4.4.3.1 Spindle Starting/Stopping Control ····44
4.4.3.2 Spindle S Command _Gear Shifting Control ····46
4.4.3.3 Spindle S_ Speed Control ····47
4.4.3.4 Servo Spindle Working State Setting ····49
4.4.3.5 Spindle Orientation Control ····50
4.4.4 Cooling Control ····53
4.4.5 Manual Tool Change Control ····54
4.4.6 Manual Tool Setting Operation ····55
4.4.7 Hydraulic Chuck Control Function ····58
4.4.8 Hydraulic Tailstock Control Function ····60
4.4.9 Other Option Functions ····62
4.4.9.1 Three-Color Indicator Control ····62
4.4.9.2 Lubricating Control ····62
4.4.9.3 Machine Electricity Delay Power-on Control ····63
4.4.9.4 External MPG Operation ····63
4.4.9.5 Safety Door Check Function ····64
4.4.9.6 Pressure Low Alarm Check Function ····64
4.4.10 Searching Run Message in JOG Working Mode ····64
4.4.11 Appendix: ····64
4.4.11.1 MDI Input Controlling M Command Table MDI ····64
4.4.12 Spindle Turn Function ····65
4.5 AUTO Working Mode ····66
4.5.1 System Working Mode in AUTO Working Mode ····67
4.5.2 Function Key Operation in AUTO Working Mode ····67
4.5.2.1 SINGLE Execution and CONTINUOUS Execution Switch ····67
4.5.2.2 Dry Run and Machining Run Switch ····68
4.5.2.3 Running a part Program from the First Block ····69
4.5.2.4 Running a Part Program From a Specified Block ····69
4.5.3 Displaying in a Part Program Running ····69
4.5.3.1 Machining Workpiece Count and Timing ····70
4.5.4 Manual Operation of Miscellaneous Function ····70
4.5.5 Speed Override Tune in AUTO Working Mode ····71
4.5.5.1 Speed Override Tune ····71
4.5.5.2 MPG Speed Control ····71
4.5.6 Interference Operation in Program Execution Process ····72

4.5.6.1 Press Key Interference in Program Execution ····· 72
4.5.6.2 External Feed/ Spindle Hold Knob ····· 73
4.5.6.3 External Start and Pause Signal ····· 74
4.5.6.4 Feed Device Alarm Function ····· 74
4.5.7 Modifying Offset in Program Run ····· 75
4.5.7.1 Modifying Offset Method in Program Run ····· 75
4.5.7.2 Modifying Tool Compensation Validity in Program Running ····· 75
4.5.8 Searching Run Message in AUTO Working Mode ····· 76
4.5.9 Program Reference Point Return in AUTO Working Mode ····· 77
4.5.10 System Reset and ESP Signal Processing in AUTO Working Mode ····· 77
4.5.11 Regulating LCD Brightness in AUTO, JOG Working Mode ····· 78
4.5.12 Display of M Command Execution State in AUTO, MANUAL Mode ····· 78
4.5.13 Operations in AUTO Mode ····· 78
4.5.14 The External Feedrate Knob in Auto, Manual Mode ····· 78
4.6 Parameter Working Mode ····· 79
4.6.1 Parameter Overview ····· 80
4.6.1.1 Parameter Privilege ····· 80
4.6.1.2 Entering Operation Level ····· 81
4.6.1.3 Parameter Management ····· 81
4.6.2 Parameter Modification ····· 82
4.6.2.1 Parameter Search ····· 82
4.6.2.2 Parameter Modification ····· 83
4.6.3 Parameter Function Key Prompt hp1 ····· 83
4.6.3.1 Parameter Communication and Standard Format ····· 84
4.6.3.2 Parameter Draw and Solidifying ····· 85
4.6.3.3 System Software Upgrade and Memory Update ····· 86
4.6.3.4 Functional Command Privilege ····· 87
4.6.4 Parameter Explanation ····· 87
4.6.4.1 Reference Point, Software Limit Bit Parameter __ P000～P020 ····· 87
4.6.4.2 Parameters Related to Zero Return Function __ P021～P026, P109～P111, P123～P125, P406～P407 ····· 88
4.6.4.3 Traverse Speed, Acceleration Time Parameter __P100～P108, P112～P119 ····· 90
4.6.4.4 Parameters Related to Transmission and Compensation__ P200～P209, P411 ····· 92
4.6.4.5 Parameters Related to Spindle, Cooling __ P300～P317, P326, P329, P341,P350, P351, P410 ····· 93
4.6.4.6 Parameters Related to Tool Post __ P318～P319 ····· 97
4.6.4.7 Parameters Related to Chuck and Tailstock __ P327～P328, P409 ····· 97
4.6.4.8 Run and Efficiency Bit Parameter __ P400～P401, P420 ····· 98
4.6.4.9 Relationship Between Path and Run, Efficiency Parameter ····· 101
4.6.4.10 Safety and Debugging Bit Parameter __ P402～P404, P419 ····· 101
4.6.4.11 Motor Drive Bit Parameter __ P405 ····· 106
4.6.4.12 Parameters Related to Other Interfaces __ P412, P330～P332 ····· 107
4.6.4.13 Auxiliary parameter __ P413～P418, P333 , P335, P027～P029 ····· 108
4.6.4.14 Interface Parameter __ P500～P566 ····· 111
4.6.4.15 Variable Initial Value __P600～P639 ····· 111
4.6.4.16 M87/M88 Related Parameter__P342, P343 ····· 112
4.6.4.17 Self-defined Related Parameter __P344~P348 ····· 112
4.6.5 Appendix: Parameter List ····· 112
4.6.5.1 Reference Parameter List ····· 112
4.6.5.2 Motion Parameter List ····· 113
4.6.5.3 Transmission Parameter List ····· 114
4.6.5.4 Auxiliary Parameter List ····· 114
4.6.5.5 Bit Parameter ····· 115
4.6.5.6 Variable Initial Value List ····· 115
4.6.5.7 Interface Parameter List ····· 116
4.6.5.8 Parameter List Related to Command Forbidden ····· 117
4.6.5.9 Parameter List Related to Output Interface Release ····· 118
4.6.5.10 Parameter List Related to Input Interface Release ····· 118
4.7 OFFSET Working Mode ····· 119
4.7.1 Tool Offset Value Search ····· 120
4.7.2 Input Tool Offset Data by Keyboard Key ····· 120

4.7.3 Tool Offset hp1 Function ····· 121
4.7.3.1 Communication of Tool Offset Data ····· 121
4.7.3.2 Clearing Offset Values of Each Group in Offset Data ····· 121
4.7.3.3 Clearing Offset Data ····· 122
4.8 Diagnosis Working Mode ····· 122
4.8.1 Interface Signal Search ····· 122
4.8.2 Interface Signal Name Display Explanations ····· 123
4.8.3 Input Interface Diagnosis Explanation ····· 123
4.8.4 Output Interface Diagnosis Explanation ····· 123
4.8.5 Output Interface Operation Function ····· 124
4.8.6 Spindle Encoder and Spindle Speed Check ····· 124
4.8.7 Diagnosis hp2 Function ····· 125
4.8.7.1 Alarm Record Display ····· 125
4.8.7.2 Alarm Record Search ····· 126
4.8.7.3 Alarm Record hp2 Function ····· 127
4.8.7.4 The Deletion of Alarm Record ····· 127
4.8.8 Machine Miscellaneous Function Control ····· 127

CHAPTER FIVE USB SYSTEM COMMUNICATION ····· 129
5.1 USB Communication ····· 129
5.1.1 USB Operation ····· 129
5.1.2 USB File Catalog Requirements ····· 129

PROGRAMMING ····· 131

CHAPTER ONE PROGRAMMING FUNDAMENTAL ····· 131
1.1 Coordinate Axis and its Direction ····· 131
1.2 Machine Coordinate System, Machine Zero ····· 131
1.3 Program Reference Point ····· 132
1.4 Machine 2nd, 3rd Program Reference Point ····· 132
1.5 Workpiece Coordinate System ····· 132
1.6 Positioning and Interpolation Function ····· 132
1.7 Programming Coordinate ····· 133
1.7.1 Absolute Coordinate Values ····· 133
1.7.2 Relative (Incremental) Coordinate Values ····· 133
1.7.3 Compound Coordinate Values ····· 134

CHAPTER TWO PROGRAM STRUCTURE ····· 135
2.1 Character ····· 135
2.2 Field ····· 135
2.3 Block Number ····· 136
2.4 Block ····· 136
2.5 Block Skip Symbol and Comment ····· 137
2.6 Program Structure ····· 137

CHAPTER THREE MSTF COMMANDS AND FUNCTIONS ····· 139
3.1 M — Miscellaneous Function (Command List) ····· 139
3.1.1 M00 — Pause ····· 140
3.1.2 M02 — End of Program ····· 141
3.1.3 M20 — End of Program Cycle Machine ····· 141
3.1.4 M30 — End of Program Spindle OFF Cooling OFF ····· 141
3.1.5 M03, M04, M05 —Spindle Control ····· 141
3.1.6 M08, M09 — Cooling control ····· 142
3.1.7 M10，M11, M12 — Clamping/Releasing Workpiece, Cancelling Chuck Output Signal ····· 142
3.1.8 M13, M14 —Spindle Clamping Signal Output and Retraction ····· 142
3.1.9 M32, M33 — Lubricating ON/OFF ····· 143
3.1.10 M41, M42, M44, M43 — Spindle Automatic Gear Shifting Control ····· 143
3.1.11 M78, M79, M80 —The Output Signal of Tailstock Forward, Backward and Cancellation ····· 144
3.1.12 M95 — Workpiece Counting ····· 144
3.1.13 M96 —Cycle Execution Call ····· 145
3.1.14 M97 — Program Transfer ····· 146
3.1.15 M98, M99 — Subprogram Call and Subprogram Return ····· 146

3.1.15.1 M99 Increase L Address Character ····· 147
3.1.16 M21, M22, M23, M24 —User Output Control ····· 147
3.1.17 M91, M92, M93, M94 — User Input ····· 148
3.1.18 M47, M48 — Set Spindle Working State ····· 149
3.1.19 M86—Send an Information Based Upon the ModBus Communication Agreement 149
3.1.20 M87, M88 — Spindle Orientation Control ····· 151
3.1.21 M50~M74 — Self-Defined Command ····· 152
3.1.21.1 M50~M74 with Field K ····· 152
3.1.21.2 In No.254 Program, When Compiling M50~M74 commands，with Parameter, but not Necessary. ····· 153
3.2 M81, M82, M83, M84—User input/output condition control ····· 153
3.2.1 M82—Output Control and Detection ····· 154
3.2.2 M81—Control According to Input Signal State ····· 154
3.2.3 M83—Control According to Output Signal State ····· 154
3.2.4 M84 — Input Signal Inspection in the Specified Time ····· 155
3.3 S function — Spindle Function ····· 155
3.3.1 Gear Shifting Controlling Spindle Motor ····· 155
3.3.2 Spindle Controlling Variable-Frequency Motor ····· 156
3.4 T function — Tool Function ····· 156
3.5 F function — Feedrate Function ····· 158

CHAPTER FOURG COMMANDS AND FUNCTIONS ····· 161
4.1 G00 —Rapid Positioning ····· 161
4.2 G01 — Linear Interpolation ····· 162
4.3 G06—Enter G06 Motion Mode with Single Axis；G07—Stop G06 Motion；G08—Enabling/Disabling G06Mode ····· 164
4.4 G09 —(Y axis) The Shifting Command in Working State of Feeder ····· 167
4.5 Thread Cutting Command ····· 168
4.5.1 G33 —Thread Cutting ····· 168
4.5.2 G34 — Variable Pitch Thread Cutting ····· 172
4.6 G32 —Tapping Cycle ····· 173
4.7 G38 — Rigid Tapping, Threading ····· 174
4.8 G50 — Setting a Workpiece Coordinate System ····· 176
4.9 G51 — Recovering Workpiece Coordinate System Setting ····· 177
4.10 G26 — X, Z, Y Reference Point Return ····· 177
4.11 G28 — Return to Machine Zero(Machine Reference Point) ····· 178
4.12 G30 — The 2nd , the 3rd Program Reference Point Return ····· 179
4.13 G04 — Timer Delay ····· 180
4.14 G96 —Constant Surface Speed Control, G97 —Constant Surface Speed Cancel ····· 180
4.15 G22, G80 —Program Part Cycle ····· 183
4.15. 1 G80 Adds L Address Character ····· 184
4.16 G98 —Feed per Minute, G99 —Feed per Revolution ····· 185
4.17 G31 — Skip ····· 186
4.18 G35 — I/O Signal Observation Command ····· 187
4.19 G52 — Rotary Axis Y Axis Coordinate Clearing ····· 188
4.20 G66—Store the Current Coordinates, G67—Return to the Stored Coordinates ····· 189
4.21 G81 — Drilling；G83 — Peck Drilling ····· 190
4.22 G0681, G0683 —The Drilling Cycle in the G06 Mode ····· 191
4.23 Appendix: G function and Its Explanation Table(table 4-2) ····· 191
4.24 Appendix： ····· 193

CHAPTER FIVE GENERAL PROGRAMMING RULES AND EXAMPLES ····· 195
5.1 General Programming Rules ····· 195
5.2 Programming Rules for Commands in One Block ····· 196
5.3 Command Execution Sequence ····· 197

CHAPTER SIX ALARM MESSAGE ····· 199
6.1 Emergency Alarm ····· 199
6.2 Alarm Table in PARAMETER, OFFSET Working Mode(i.e. E001~E009) ····· 199
6.3 General Chart of Alarm in Edit Working Mode(i.e. E100~ E199) ····· 201
6.4 Emergency Alarm Program Alarm Table (i.e.E200~ E299, E600~ E699) ····· 202
6.4.1 Alarm in Program Command (i.e. E200~299) ····· 203
6.4.2 Alarm in Program Compound Check (i.e. E600~699) ····· 205

6.5 Alarm Table in JOG OR AUTO Working Mode (i.e.E300~ E499) ····206
6.5.1 Alarm in Executing Relative Operations (i.e E300~E399) ····207
6.5.2 Relative Alarm in Executing Statement (i.e.E400~ E499) ····211

CHAPTER SEVEN STATEMENT PROGRAMMING ····213
7.1 Variable ····213
7.1.1 Variable Expression Method ····213
7.1.2 Classification of Variable ····213
7.1.2.1 Common Variable ····213
7.1.2.2 Pointer Variable ····216
7.1.2.3 Interface Variable ····217
7.1.2.4 Keyboard Scan Register r5001 ····218
7.1.2.5 Display Window Register r5002 ····219
7.1.2.6 Display Value Register r5003 ····222
7.1.2.7 Graph Update Register r5004 ····222
7.1.2.8 Program Control Register r5008 ····223
7.1.2.9 System Special Variable Set 1 ····224
7.1.2.10 System special variable set 2 ····224
7.2 Statement ····227
7.2.1 Assignment Statement ····227
7.2.2 Conditional Statement ····228
7.2.3 Statement Program Example ····229
7.3 Process Monitoring and Execution ····230
7.3.1 Process Monitor Description (r7000) ····231
7.3.2 Start and Close of Process Monitor ····232
7.3.3 Monitor Program Example ····234
7.3.4 Multi-statement Global Monitoring Function ····235
7.3.5 Pulse Monitoring (r7100) ····236
7.3.6 Pulse Monitoring Program Example ····238
7.3.7 Variable Transfer Register (r7900) ····238
7.4 Attached List ····239
7.4.1 ASCIIList ····239
7.4.2 Often Used Color and Code Value Corresponding List ····240

CHAPTER EIGHT CUSTOMIZATION COMMAND PROGRAM ····241
8.1 Customization Command ····241
8.1.1 Customization Command Program Format ····241
8.2 Customization Command Store (P254) ····242
8.2.1 Format and Debugging of Customization Command Storeroom ····242
8.2.2 Explanation of Customized Command Storage ····243
8.2.3 Customized Command Machining Example ····244
8.3. Foot switch in M61 command ····247
8.4 The Foot Switch of M51~M58 Commands ····247
8.5 Call M50-M72 Function Before Machining ····248
8.6 Call M50-M70 Function by Manual 0 Key ····248

CONNECTION ····249

CHAPTER ONE INTERFACE ····249
1.1 Rear Cover Interface Position Layout ····249
1.2 Total Frame ····250

CHAPTER TWO INTERFACE GRAPH ····251

CHAPTER THREE CNC DEVICE CONNECTION ····253
3.1 Front Cover Communication Interface ····253
3.1.1 USB Interface ····253
3.2 X1, X2 Interface ····253
3.2.1 X1, X2 Interface Signal Definition ····253
3.2.2 Connection Method of Input Signal ····256
3.2.3 Connection Method of Output Signal ····259
3.2.4 Input/Output Signal Technical Specification ····260
3.3 Machine Zero Return Function and Connection ····260
3.4 Tool-Exchange Control Function and Connection ····262
3.4.1 Tool-Exchange Control Signal Definition ····262

3.4.2 Signal Connection ····· 262
3.4.3 Function Description ····· 263
3.4.3.1 Tool Change Mode 0 ····· 263
3.4.3.2 Tool Change Mode 9 ····· 263
3.5 X3 Motor Interface ····· 265
3.5.1 Signal Definition ····· 265
3.5.2 Technical Specifications ····· 265
3.5.3 Equivalent Circuit ····· 265
3.5.3.1 Drive Unit Alarm Signal XALM, ZALM, YALM ····· 265
3.5.3.2 Enable Signal Xen, Zen, Yen ····· 266
3.5.3.3 Pulse Signal and Direction Signal ····· 266
3.5.4 Connection Between CNC System and Drive Unit of Compound Stepper Motor ···· 267
3.5.5 Connecting Between CNC and Drive Unit of Reaction Stepper Motor ····· 269
3.5.6 Connection Layout Between CNC and AC Servo Drive Unit ····· 271
3.5.7 Connection Layout Between CNC and Panasonic Drive Unit ····· 273
3.5.8 Connection Layout Between CNC System and Japanese Yaskawa Drive Unit ····· 274
3.6 X4 Spindle Interface and Y interface ····· 275
3.6.1 Signal Definitions ····· 275
3.6.2 Converter Technical Specification ····· 276
3.6.3 Encoder Technical Specifications ····· 276
3.6.4 Connection Layout of Converter Analog Voltage ····· 277
3.6.5 Encoder Interface Method ····· 277
3.6.6 Encoder Interface Connection Layout ····· 277
3.6.7 Connection Between CNC System Y and AC Servo Drive Unit ····· 278
3.6.8 Connection Between CNC System Y and DAP03 Spindle Drive Unit ····· 279
3.7 X5 MPG Interface ····· 280
3.7.1 Signal Definition ····· 280
3.7.2 Interface Circuit Principle ····· 280
3.7.3 Connection Layout ····· 280

CHAPTER FOUR USE AND MAINTENANCE ····· 281
4.1 Environmental Condition ····· 281
4.2 Earthing ····· 281
4.3 Power Supply Requirements ····· 281
4.4 Guard ····· 281
4.5 Reuse it Again ····· 281

APPENDIX ····· 283

Appendix 1 CNC system electrical component symbol explanations ····· 283
Appendix 2 CNC system tool post controller circuit method layout ····· 284
Appendix 3 Interface circuit method layout ····· 285
Appendix 4 External control connection layout ····· 288
Appendix 5 CNC system appearance installation dimension ····· 289

OPERATION

CHAPTER ONE OVERVIEW

GSK96 is a newly-developed multi-function position control system with the precision control function of μm level for Z/X/Y axis. The single-axis, double-axis or three-axis control can be set by parameters. This system can realize motion functions such as positioning and feeding, two-axis or three-axis linkage, taping and threading, rotation indexing and servo spindle control, and has plenty of input/output signal control functions, which can realize complicated control of multi-signal detection and output.

GSK96 control system is applicable to the control like indexing drilling, grinding, taping and feeding, cutting and welding rather than the control of the turning machine.

This product is easy to operate and has a visible lattice true color LCD interface with resolution of 800X480. It is characterized by its powerful functions, stable performance, full-screen program editing, Chinese operation interface, outstanding safety, machining precision and machining efficiency as well as the high cost performance. In addition, it employs the international standard CNC language ISO code to write programs.

Technical Specifications:

- ✓ **X, Z Y single-axis motion, which can realize the linkage of any two-axis, three-axis linkage; interpolation precision: 0.001mm; the Max. rapid traverse speed: 30m/min**
- ✓ **Any of X, Z, Y axis can be set as the rotation axis control**
- ✓ **Control servo spindle**
- ✓ **Flexible and convenient programming, with statement programming function**
- ✓ **USB interface communication to get the convenient and fast operation**
- ✓ **Least command unit 0.001mm, command electronic gear ratio (1～9999999)/(1～9999999)**
- ✓ **Automatic gear shifting of all kinds of spindle**
- ✓ **Backlash compensation, tool length compensation function**
- ✓ **Tapping function**
- ✓ **Course monitoring function**
- ✓ **Cutting metric/inch straight thread, variable pitch thread**
- ✓ **Full-screen part programs editing, capable of storing 255 machining programs; No. 253 program up to 4MB**
- ✓ **Big screen color LCD, color configuration is selected by the parameter**
- ✓ **MSTF state real-time display in machining**
- ✓ **Multi-level operation password for convenient device management**
- ✓ **Parameter backup function**
- ✓ **Parameter, offset data communication function**
- ✓ **Bilateral communication between CNC and USB, CNC is upgraded by USB**

CHAPTER TWO TECHNICAL SPECIFICATIONS

2.1 GSK96 Technical specifications

Motion control	Controlled axes: X, Y, and Z; The three axes are respectively set as the linear axis or rotation axis control; simultaneous controlled axes (interpolation axes): 2-axis linear interpolation or 3 axes (X, Y and Z).
	Interpolation: X/Z, Z/Y or X/Y two-axis linear interpolation，X/Z of any single-axis screw thread interpolation or X/Y/Z three-axis linkage.
	Position command range:-9999.999 mm～9999.999mm; least command unit: 0.001mm
	Electronic gear: Command multiplier coefficient 1～9999999，command division coefficient 1～9999999
	Rapid traverse speed：up to 30000mm/min; rapid override：F25%, 50%, 75%, 100% real-time regulation
	Cutting feedrate: up to 15000mm/min; feedrate override：0～150% 16 grades real-time regulation
	MANUAL feedrate:： 0mm/min～1260mm/min 16-grade real-time regulation or it is defined in real time
	MPG feed：0.001mm, 0.01mm, 0.1mm
G command	G commands: G00, G01, G06, G07, G08, G09,G04, (G22/G80), G26, G28, G30, G31, G32, G33, G34,G35, G38, G50, G51, G52, G66, G67, G81, G0681, G83, G0683, G96, G97, G98,G99
Thread machining	Capable of machining single/multiple straight thread, variable pitch thread; pitch: 0.001mm～500mm or 0.06tooth/inch～25400teeth/inch; with tapping function
	Spindle encoder: lines can be set within 100p/r～5000p/r; Drive ratio between encoder and spindle is 1:1
Precision compensation	Backlash compensation: 0 mm～10.000mm
	Tool offset: 16 tool numbers, 64 groups of tool length compensation; Tool setting method: trial cutting; Tool offset can be modified during program execution, and can be modified by statement command
M command	M00, M02, M20, M30, M03, M04, M05, M08, M09, M10, M11, M12, M13, M14, M32, M33, M41, M42, M43, M44, M47, M48, M86, M87, M88, M78, M79, M80, M81, M82, M83, M84, M96, M97, M98, M99, M91, M92, M93, M94, M95, M21, M22, M23, M24； The user-defined M commands: M50～M74, which can realize the special function control
T command	Up to 16 tools，setting tool post type parameters to select the control process of tool change (the system has no the integrated function to control the electric tool post on the machine) Tool change action is not executed if the tool post type is set to 0, and the system calls M60 command to execute tool change when set to 9.
Spindle speed control	Speed switching value control: S 4-gear directly controlling output range is S01～S04; or 16-gear BCD output range is S00～S15
	Speed analog voltage control: S specifies the spindle speed per minute or the cutting surface speed (constant surface speed) , outputs 0～10V voltage to spindle converter, supports 4-gear spindle speed M41～M44 with stepless shifting gear
	Support for the speed/position control mode switch for DAP03 servo spindle, which realizes Z or X axis linkage function
I/O function	I/O function diagnosis display
	I/O interface：23 input/18 output interfaces
Statement programming	Assignment statement: complete assignment, many arithmetic and logic operations
	Conditional statement: complete conditional judgment and skip
Display window	Display: 800x480 lattice, color LCD，with LED or CCFL backlight
	Display method: Chinese or English window set by a parameter
Program edit	Program capacity: max. 255 programs with 4400KB
	Edit method: edit in full screen, relative/absolute coordinate compound programming, support for subprogram call and subprogram multi-level embedding
Communication	USB interface; bidirectional transmission of programs, parameters and offset between CNC and USB; Support for USB to download and upgrade
Optional drive unit	DA98 Series Digital AC Servo or DY3 Series Stepper Drive unit with pulse + direction signal input

CHAPTER THREE OPERATION PANEL

GSK96 multi-function position control system (abbreviated to system hereafter) employs the aluminum alloy operation panel.

3.1 LCD Display

LCD display: CNC man-machine dialogue interface. Lattice color LCD display with resolution of 800x480.

3.2 LED Status Indicator

LED indicator indicates the current working state of the system. There are 14 function keys with LED indicators. The function executed by the corresponding key is valid when LED is ON, and it is invalid when LED is OFF.

3.3 Keyboard

Based on GB/T 3168-1993 Numerical Control of Machine-Symbol, the system sets the following symbol function keys that complete the corresponding functions when they are pressed as follows:

3.3.1 Character Keys

Character keys include numbers, letter, and some other symbols.

In EDIT working mode, each letter key can switch into 2 or 3 letter keys; in other working mode, each letter key only expresses one letter key. (For example, I and P are on one key, operator directly presses the key when “I” or “P” is required, and the system automatically indentifies other letters.)

Numeric keys: input data(0～9) ;

Letter keys: input letters;

Symbolic keys: input + (plus), - (minus), *(multiply), / (divide), + (positive), - (negative), (decimal point),

Logic keys:＞(larger than), ＝(equal to), ＜(smaller than), and, or, (), etc.

3.3.2 Working Mode Selection Key

Press the symbols and characters and the working mode selection keys to complete the corresponding function, and their definitions are as follows:

: select EDIT working mode

: select JOG working mode

: select AUTO working mode

: select PARAMETER working mode

: select OFFSET working mode

: select DIAGNOSIS working mode

3.3.3 Function Keys

Press function keys with the visualization symbol and letter to complete the corresponding functions and each symbol definition is as follows:

INCREASING RAPID OVERRIDE Increase rapid traverse override in JOG working mode and G00 rapid traverse override in AUTO working mode.

REDUCING RAPID OVERRIDE: Reduce rapid traverse override in JOG working mode and G00 rapid traverse override in AUTO working mode.

INCREASING FEEDRATE OVERRIDE: Increase feedrate override in JOG working mode and G01 feedrate override in AUTO working mode.

REDUCING FEEDATE OVERRIDE: Reduce feedrate override in JOG working mode and G01 feedrate override in AUTO working mode.

X, Z, and Y PROGRAM REFERENCE POINT RETURN: It is valid in JOG /AUTO working mode. (Program reference point is also called program zero point in the user manual.)

X, Z, and Y MACHINE REFERENCE POINT RETURN: It is valid in JOG working mode. (Machine reference point is also called machine zero point in the user manual.)

Dry run key When Dry Run is selected in AUTO operation mode to execute commands, whether M, S, T commands are valid is determined by bit parameter **P401_d7**. After the Dry Run state is exited, the coordinate of each axis of the system automatically resumes to the one before Dry Run.

Single/Continuous key selects Single/Continuous mode in AUTO operation mode.

3.3.4 Cycle Start and Cycle Pause (Feed Hold) Key

Start and pause programs in AUTO working mode and each key symbol definition are as follows:

CYCLE START: Start to run programs in AUTO working mode; move coordinate axis in JOG working mode.

CYCLE PAUSE (FEED HOLD): pause the running in JOG or AUTO working mode;

3.3.5 Manual Axis Control Key

Manual key symbol definitions in JOG working mode are as follows:

X/Y/Z axis moves negatively in JOG working mode.

X/Y/Z axis moves positively in JOG working mode.

RAPID TRAVERSE/FEED Switching rapid traverse and feed in JOG working mode.

JOG STEP Select each step width or MPG feed in STEP/ MPG(Handwheel) working mode; **hp** function in other working modes.

MPG (Handwheel) MPG control selection and axis selection in JOG working mode; **hp** function in other working modes.

X/Z/Y and MPG axis selection in JOG working mode.

STEP/JOG mode Switch STEP/JOG mode in JOG working mode.

3.3.6 Manual Auxiliary Function Key

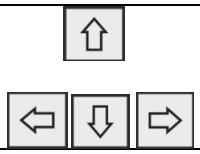
The following press keys are used to control and complete all miscellaneous function of the machine and each key symbol definition is as follows:

Spindle rotation (CW) Spindle rotates clockwise. (View from tailstock to chuck along the spindle)

Spindle stop Spindle stops.

Cooling control Switch cooling ON/OFF.

3.3.7 Edit Keys

Press key	Name	Function explanation
	ENTER key	Press it after the corresponding operation is performed.
	INPUT key	Input the required content.
	ALTER key	Switch character insert/alter state in EDIT working mode; Special definition in other working modes.
	DELETE key	Delete character, letter, block or whole program in EDIT working mode; Special definition in other working modes.
	ESCAPE key	Cancel the current input data or exit from the working state; exit from the current operation or setting.
	HOME key	“DRY RUN” in AUTO working mode; Cursor moving to the beginning of the current line in EDIT working mode.
	END key	“STEP” in JOG working mode; Cursor moving the end of the line in EDIT working mode.
	Help key	Two help keys: hp1~hp2 When step adjustment is invalid, hp2 is valid.
	Cursor movement key	Control cursor movement in EDIT/PARAMETER/OFFSET working mode; **Hp** function or other special definitions in other working modes.
	PAGE UP/DOWN	Display page up/down in EDIT/PARAMETER/OFFSET; Special definition in JOG /AUTO working mode.

3.3.8 Reset Key

.Press key	Name	Function explanation
	RESET key	Reset

CHAPTER FOUR SYSTEM OPERATION

This chapter introduces operations and notes of the system. Please read carefully before operation.

4.1 System ON/OFF, Initial State, Modal, and Safe Protection

4.1.1 Power On

There is no power switch on the operation panel of the system. The operator installs it according to the different machine to avoid bad effects to CNC system owing to the impaction of power supply.

Check before the system is turned on:

1) Ensure the machine state is normal;

2) Ensure the voltage meets the requirements;

3) Ensure the wiring is correct and firm.

The system is turned on as follows:

1) The master power switch of machine is turned on.

Switch on the power switch of the CNC system, and the system displays as Fig. 4-1. Press any keys, and the system enters into EDIT working mode.

Fig. 4-1 System initialization display window

2) The system orderly completes the following work after power-on:

- The system controls the program loading.
- The system automatically checks itself and executes the initialization.
- The system loads and checks parameters.
- I/O interface initialization.
- The system loads and checks the operator programs.

【Note】

1) Must not press any keys on the system panel when the system is turned on. Press RESET key when the system enters the press key test window at the moment.

4.1.2 Power Off

The system is turned off as follows:

1) The power switch of the CNC is turned off.
2) The power switch of the machine is turned off.

Check before the system is turned off:

1) X, Z, Y are in the stop state;
2) Miscellaneous function(spindle, cooling) OFF;
3) Turn off the power supply.

【Note】

1) The system should be checked itself and initialized when it is turned on at first (it is completed by the machine manufacturer, and the operator cannot execute the operation, otherwise, the parameter set by the machine manufacture will lose).
2) Operations related to turn off the machine power supply are referred to the operator manual machine manufacturer.

4.1.3 System, Program Initial and Modal

4.1.3.1 The Initial State and Modal of the System

The initial mode of the system is defined to be a special initial state of all functions set by itself when the system is turned on; all auxiliary functions do not execute the actual output.

The modal of the system is defined to be their kept states after the system executes all functions. Initial mode and modal of the system:

System state	Initial mode	Modal
Machine coordinate system of the system	Keep last power-on state	Keep till being changed
Tool nose coordinate system of the system	Keep last power-on state	Keep till being changed
Cutting feedrate：F	In Auto mode：30mm/min	Keep till being changed
	In JOG mode: Keep last power-on state	
Frequency conversion spindle speed：S	S200	Keep till being changed
Spindle gear	Shifting gear spindle gear：S0	Keep till being changed
	Conversion spindle gear：M41	
MANUAL slow feed/rapid feed state	Slow feed	Keep till being changed
Feedrate override	Keep last power-on state	Keep till being changed
Rapid override	Keep last power-on state	Keep till being changed
Spindle state	M05 spindle stop	Keep till being changed
Cooling state	M09 cooling OFF	Keep till being changed
Chuck state	M11 chuck release	Keep till being changed
Lubricating state	M33 lubricating OFF	Keep till being changed
T number state	Keep last power-on state	Keep till being changed
Tailstock state	M79 tailstock run-out state	Keep till being changed
Set spindle speed/position mode	M48	Keep till being changed

4.1.3.2 Initial Mode and Modal of Program

The initial mode is the automatic initialization setting state before the system executes the machining program; i.e. the initial default state of the default programming word and speed word.

Program initialization state of the system:

G command：G00, G97, G98;

Cutting speed：30mm/min;

Miscellaneous function: current state;

System coordinates: current coordinates are those of the last automatic executing program or manual operation

G modal is always valid till it is changed by other modal commands in the same group after the word is set. After the modal meaning is set, the G command may not be input again when the same function is used in the later block.

There are three groups of G command with modal characteristics, and there is only one command of each group in the initial state:

Group 1：G00, G01; (initial mode：G00) ;

Group 2：G96, G97; (initial mode：G97) ;

Group 3：G98, G99; (initial mode：G98 F30) ;

The command without modal characteristics has effect in the block and must be defined to use every time.

【Note】

In AUTO working mode, the system automatically recovers to the program initial mode when it executes the first command of workpiece program or executes the first block command after M20, or selects the middle block as the first command.

4.1.4 Safe Protection

The CNC system sets a perfect protection measure to prevent the operator from danger and the machine from being damaged.

4.1.4.1 Hardware Limit Protection

The system can check the travel limit switch installed on the machined. When the machine slide moves to press the travel limit switch, the system stops feeding instead of closing other miscellaneous functions, and the program stops running and the system displays the hardware limit alarm information of corresponding axis.

After the travel limit switch alarms, the system can select JOG working mode, the key for axis movement which is reversed to the limited direction is pressed, i.e. the system exits the travel limit and the travel limit switch alarm automatically disappears on the screen.

【Explanation】

1）X, Y, Z positive limit check shares one pin +LT, and their negative limit check shares one pin -LT; when the positive limit alarms, all axes cannot move positively but move negatively; and vice versa.

2）When the travel limit switch runs across the limit block, the limit signal appears; the valid length of limit block signal is more than 30mm or more to avoid rush out the valid area of the signal.

3）When the parameter is set to “limit emergency stop” mode (bit parameter **P402_d7=1),** and the system runs across the limit block, there may be great deviation between the coordinates displayed by the system and the actual position. Adjust the machine coordinates.

【Relative parameters】

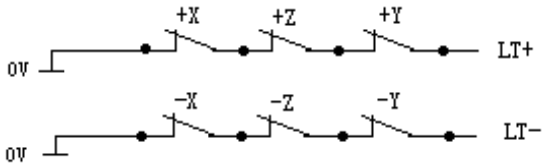
Bit parameters: P402_d7, P404_d6, P404_d1.

Bit parameter **P402_d7** sets the hardware limit alarm mode;

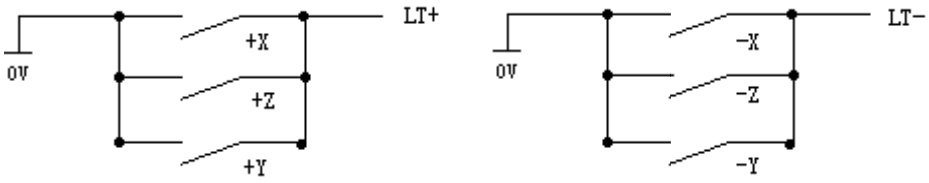
Bit parameter**P402_d6**sets whether the hardware limit alarm checks;

Bit parameter**P402_d1**sets the hardware limit alarm level of each axis;

When **P404_d1=1** is high level alarm, the positive limit switch +X, +Y, +Z of each axis are normally closed contact, are connected to X/Z/Y positive limit input interface +LT (they are open and the system alarms) in serial; the negative limit switch -X, -Y, -Z of each axis are normally closed contact, are connected to X/Z/Y negative limit input interface -LT (it is off and the system alarms) in serial; it is suggested that the operator should select in prior the hardware limit to the normally closed contact of each axis as follows:



When **P404_d1=0** is low level alarm, the positive limit switch +X, +Y, +Z of each axis are normally open contact, are connected to X/Z/Y positive limit input interface +LT (they are closed and the system alarms) in serial; the negative limit switch -X, -Y, -Z of each axis are normally open contact, are connected to X/Z/Y negative limit input interface -LT (it is off and the system alarms) in serial; the connection is as follows:



4.1.4.2 Software Limit Safe Protection

1）Mechanical software limit safe protection

The mechanical software limit safe protection is to limit machine coordinate motion range to avoid slide to exceed the motion range. The mechanical software limit alarms when the machine position (machine coordinates) exceeds the range.

Releasing overtravel alarm methods: reversely movement in JOG working mode (negatively

moves for positive overtravel; positively moves for negative overtravel).

2） Tool nose software limit safe protection

The tool nose software limit safe protection is to limit tool nose coordinate motion range to avoid the tool nose to exceed the motion range. The tool nose software limit alarms when the machine position (tool nose coordinates) exceeds the range.

Releasing overtravel alarm methods: reversely movement in JOG working mode (negatively moves for positive overtravel; positively moves for negative overtravel).

【Explanation】

1) The coordinate axis decelerates to stop when the coordinates exceed the software limit range during the motion.

【Relative parameters】

P009, P010: max. travel of Z positive, negative tool nose software limit; **P011, P012**: max. travel of X positive, negative tool nose software limit;

P013, P014: max. travel of Y positive, negative too nose software limit; **P015, P016**: max. travel of Z positive, negative mechanical software limit;

P017, P018: max. travel of X positive, negative mechanical software limit; **P019, P020:** max. travel of Y positive, negative mechanical software limit;

Bit parameter **P404_d4, P404_d3** separately sets whether the mechanical, tool nose software limit alarm are valid.

4.1.4.3 ESP Alarm (Emergently Stopping the System)

When there is the external emergency stop input terminal ESP in the system input interface, the operator should connect the emergency stop button Normally closed contact on the machine panel with the emergency stop input terminal. Press ESP Button, then the system enters the emergency stop state. The system stops all feed and sets the output of all the switch value control such as the spindle and the cooling to be invalid, displaying “ESP Alarm”. (If the system interface is sprung out other window by chance, and then perform the ESP function in advance, the “ESP alarm” will display later).

After the emergency stop condition is released, the operator should rotate the emergency stop button clockwise based on the up arrow, the button automatically releases to cancel the emergency stop signal.

When the system is in the emergency stop alarm state and the external emergency stop signal is cancelled, you can press the “RESET” key to return the previous working mode.

When the system is in the emergency stop state, if the external emergency stop signal is not cancelled, it is forbidden to operate the system in Manual, Auto and Diagnosis operations; but the user can press RESET key to remove the alarm window, and switch the control to Edit, Parameter or Tool Offset operation mode.

If the emergency signal is not cancelled in Edit, Parameter or Tool Compensation operation, press RESET key to remove the alarm window, then it is available to operate system.

There is the ESP in the movement, there may be great deviation between the coordinates displayed the system and the actual position, and the operator must correct the machine coordinates.

When emergency button is pressed, there are a series of procedures to be handled:

1）During emergency stop, the system stops all feed, and terminates program execution; the spindle stops rotating; cooling and lubricating function is turned OFF.

2）During emergency stop, the system automatically sets the internal stored chuck and tailstock states to M11 and M79. After the emergency is released, when the foot switch of chuck or tailstock is pressed for the first time, the states become M10, M78.

3）During emergency stop, if such commands as tool exchanging, tailstock, chuck, gear changing is being executed, the execution should be stopped immediately (cancel the tool post CCW/CW signal; tailstock signal and chuck signal are depends on parameter setting); at this time, the system assumes that the tool post, chuck, tailstock and gear are not in the proper position, and red indicator is flickering; only when the emergency stop alarm is released, and above execution is repeated or after system power-on, can the system resume the normal state.

4）During emergency stop, if the output signal of MDLY interface remains the same, interface control signals other than MDLY, spindle , cooling, lubricating are set by parameters. If P403_d3 is 0, then, only output signals for spindle, cooling and lubricating are turned off; if P403_d3 is 1, all interface control signals including chuck and tailstock signals are turned off.

5）After the emergency stop alarm is cleared, in AUTO mode, the system will exit from dry run state if it was; in MANUAL mode, the system will turn from rapid traverse state to feed state; if the set F value is not changed, the spindle analog voltage output remains the same; except for that, other functions of the system are in initial state.

6）After the emergency stop alarm is released, time counting of the low-pressure detection function and auto-lubricating control function are restarted.

【Special Attention】

1）The standard emergency stop function is actually to set the output signals to “ON” or “OFF”. It can be like this: In MANUAL/AUTO mode, when emergency stop button is pressed, the standard emergency stop is executed, in addition, M74 user-defined command is also executed (it is executed only when there is already a M74 command in the system). This function is applicable to some special machine parts under the requirements of keeping some output signals ON while turning OFF some others. If any similar alarm occurs during the execution of M74, the execution stops. During emergency stop, if the M74 command is executed, and axis movement or tool change is being carried out, the system will stop M74 execution automatically.

2）In MANUAL/AUTO mode, when the parameter sets that M74 can be executed during emergency stop (**P412_d1=1**), the pop-up window displays “+M74”. (When there is already a solidified M74 command in the system.)

3）For more details about M74, please refer to Chapter 8.

4）Please be careful when executing M74 function. It is only suitable for some special machine tools.

【Relative parameters】

The external emergency stop signal function is valid when **P404_d7** is set to 0.

The external emergency stop signal function is invalid when **P404_d7** is set to 1.

P403_d3=0：When the emergency alarm is valid, the system only turns off the signals of spindle, cooling and lubrication.

P403_d3=1：When the emergency alarm is valid, the system turns off all the output signals of the auxiliary function.

P404_d7 is for debugging the system, and must be set to valid in the power-on state; otherwise, it cannot provide protection.

P412_d1 parameter sets whether to execute M74 self-defined command during emergency stop.

4.1.4.4 Drive Unit Alarm

When the system is connected with the alarm output signal of a drive unit and appears Drive unit alarms, the system automatically closes all feed and prompts Z/X/Y drive unit alarms. All axes immediately stop motion, and the program stops running. At the moment, check the drive unit and relative device to troubleshooting and the system is turned on again.

When there is the alarm in the course of motion, there may be great deviation between the coordinates displayed the system and the actual position, and the operator must correct the machine coordinates.

In JOG working mode, the system prohibits all axes moving when there is the alarm.

In AUTO working mode, the system prohibits the program starting run when there is the alarm.

【Relative parameters】

The drive unit alarm checks when **P404_d5** is set to 0.

P405_d4, P405_d3, P405_d2 separately sets alarm level of Z, X, Y drive unit.

4.1.4.5 Other Alarms

When the system appears other alarms, Chinese characters prompts, at the moment, the operator can perform the troubleshooting based on **PROGRAMMING, Chapter 6 Alarm Message**.

4.1.4.6 Switching off Power Supply

The machine power supply is switched off immediately to avoid the accidence in the danger or other emergency cases when the machine is running.

Note: When the coordinate axis is moving and the power supply is switched off, after the machine is switched on again, there may be great deviation between the displayed coordinates and the

actual position, and so the operator must execute the machine zero return or other ways to regulate the machine coordinates to ensure that the displayed coordinates are the same as those of the actual.

4.1.4.7 Reset Operation

When the system outputs abnormally and the coordinate axis moves abnormally, the operator should press

to make the system be the reset state as follows:

1) All axis motions decelerate to stop;

2) Whether M function (spindle, cooling) output is valid is set by **P403_d2.**

3) The automatic run ends, the modal function and state keep.

4) When the system is in G00，G97，G98 state, both F value and spindle analog voltage output keep unchanged.

5) The system terminates the operation in progress, and returns the control to the initial interface of the current operation mode.

[Special attentions]

1) The standard Reset function executed by the system is the function of turning on or turning off the system output signals uniformly. The following function can be set: In Manual/Auto operation, after Reset button is pressed, and the system executes the standard Reset function, the system executes an additional M73 custom command (only when in Manual/Auto operation mode and when there is a solidified M73 command in the system, the execution is available). This function is applicable to some special machine components and is used when only some of the output signals are required to turn off and some maintained. If there is an alarm occurs during the execution of the M73 custom command, the execution is stopped.

2) In Manual/Auto operation mode, when the M73 custom command can be executed in ESP state by setting a parameter (P412_d2=1), there is a prompt “+M73” (indicating the M73 is being executed) added on the reset window (if there is a solidified M73 custom command in the system).

3) For the programming, debugging and fixing of M73 custom command, please see Chapter Eight “Custom Command Programming” in this manual.

4) Be careful to use M73 custom function during Reset. The function is only applicable to some special machine tools.

【Relative parameters】

P403_d2=0: the system closes M03, M04, M08, M32 output signals when the reset key is pressed.

P403_d2=1: the system does not close M03, M04, M08, M32 output signals when the reset key is pressed.

Parameter **P412_d2** sets whether M73 custom command is executed during Reset.

4.2 CNC Working Mode Selection

The system uses the working mode key to directly select the overall working modes. All working modes can direct switch to get the simple and convenient operations.

The display is as Fig. 4-1 after the system is turned on, and the display state keeps till the other key is pressed to enter the EDIT working mode.

4.3 EDIT Working Mode

EDIT working mode: it is the working mode to execute the part program by the operation panel. There is the corresponding intelligent prompt message for each operation. At the same time, the operator can press the prompt key- hp1 at the top right to learn the operation key list in EDIT working mode.

The relative setting or operation key format and sample descriptions in the user manual are as follows: meanings and uses of all required functions are described at the beginning of the corresponding chapter; all required letter and digit keys are expressed with underlines; the system prompt messages are expressed with borders.

In executing some setting or input or man-machine dialogue, press ESC key to exit the current operation before ESC key is not pressed.

◆ Main functions in Edit mode include:

☆ select, create, rename, copy and delete part programs;

☆ input, insert, modify and delete the content of the selected part program;

☆ transmit part programs between U disc and the system by the **USB** interface;

☆ compile and save program;

☆input variable and macro character string.

Press EDIT to enter the EDIT working mode. The EDIT working mode includes two main windows: program catalog search window and program edit window. The program catalog search window is as Fig. 4-2:

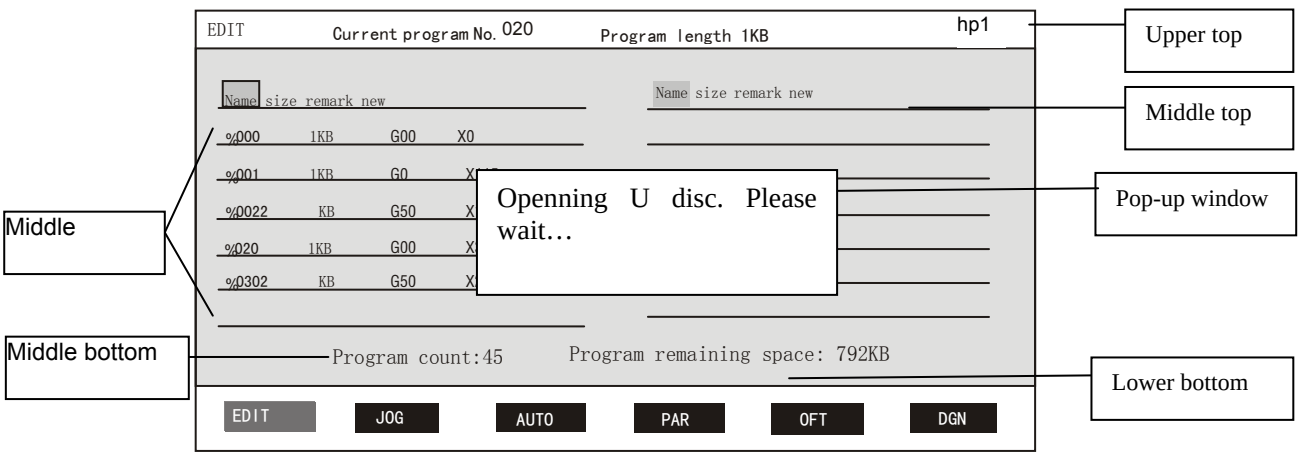


Fig. 4-2 program catalog search

◆ Display content in window area:

Upper top: Program number and capacity of current program (program length), the system function operation method prompt key hp1;

Middle top: The system will arrange the program sequence according to the name, size, remark and new);

Middle: Display program name, capacity and remark;

Middle bottom: Operation prompt message;

Lower bottom: Display current stored part program quantity (up to 255) and surplus stored space;

Pop-up window: Display operation prompt message.

【Note】

Press **hp1** key, and the system prompts “Program catalog window message prompt”, introducing the used press key functions.

4.3.1 Part Program Catalog Search

Program catalog search window displays the current stored program quantity, and all programs sorts as follows:

1) Name: program number from top to bottom, from left to right;

2) Size: program stored space from top to bottom, from left to right;

3) Remark: the first 12 characters of the first line of the program from top to bottom, from left to right;

4) The latest: Input time sequence from top to bottom, from left to right.

Press , and , , the system searches all part programs, displaying 12 program on each window; press to the first page of program, to the last. Press , to sort all programs according to their name, size, remark and last distribution.

4.3.2 Selection, Creation, Deletion, Rename and Copy a Part Program

Select, create, delete, rename and copy part programs

【Note】

1）The system executes most %000～%254 programs, 255 program names. The system prompts **E160 input program error** when the executed program name is more than 254.

2）There is no part program or the system is not used firstly; the system automatically creates and selects %000 program as the current program. When there are programs, the system sorts the program according to part program quantity and program names in the last

power-off.

3）The system supports many input, the leading zero cannot be input. Example, inputting %003 program. Press INPUT, input 0 0 3; also input 03; or input 3.

4.3.2.1 Selecting and Creating a Part Program

The operations to select a part program or create a new program are as follows:

① Press INPUT key in EDIT working mode;

② Input the required program number by the board key, or input a program number which is not in the program catalog list as the new program number;

③ Press ENTER key;

④ Select or create a part program, display the content of the part program, and the system enters the program edit window.

【Note】

1) When a program is selected, it is changed by the above steps; it cannot be changed once it is confirmed.

2) When a program number which is not exist in the part program catalog, the newly created program is taken as the current program.

【Example】

Example 1：There is no %20 part program in the part program catalog, creating it is as follows: Press keys to input: INPUT 2 0 ENTER. The new program %020 has been created and the system enters %020 program edit window.

Example 2：There is **%001** part program in the part program catalog, select it is as follows: Press keys to input: INPUT 1 ENTER. The selection of **%001** part program is completed, the system enters **%001**program edit window.

4.3.2.2 Delete a Part Program

Delete a part program is as follows:

① Press INPUT key in Edit working mode;

② Input the required program number to delete by the key board;

③ Press DELETE key, the system prompts: Enter-confirm the deletion Esc-escape the deletion .

④ Press ENTER to delete the part program which program number is input. Press ESC not to execute the deletion operation and return to EDIT working mode.

【Note】

1) The system displays **E100 the program to be deleted does not exist** when there is no the program to be deleted;

2) When the program which is to delete exists: when it is not the current program, the system deletes the program from the program list; when it is the current program, the system deletes

the program from the program list, and searches the program which program number is the smallest to be the current program; when there is no programs, the system creates one program number 000 as the current program.

【Example】

Example: delete %003 operation is as follows:

Input by the press key: INPUT 3 DELETE ENTER. The %003 program is deleted from the components program storage area.

4.3.2.3 Deleting All Part Programs

Clear the program area in the program catalog search window, and all programs in the system are deleted as follows:

① Press INPUT in the part program catalog search state;

② Input – +, 0 by the key board;

③ Press DELETE, the system prompts: Enter-delete all programs Esc-escape the deletion .

④ Press ENTER to delete all part programs; press ESC not to execute the deletion operation and return EDIT working mode.

【Note】

The system creates a program number 000 as the current program after all part programs are deleted.

P416_d6 sets whether all the programs can be deleted in EDIT operation mode.

4.3.2.4 Renaming a Part Program

The program name of the current program is renamed as another new one. The new is taken as the current program is as follows:

① Press INPUT key;

② Input the program number which is not in the program list, press ALTER and the current program number is rewritten to the input program number.

【Note】

When the input program exists, the system prompts **E166 required renaming program already exists**.

【Example】

Example: the current program %000 is renamed to %005 as follows:

Press key input: INPUT 5 ALTER. And the renaming is completed.

4.3.2.5 Copy a Part Program

Copy the current program content to another one new and the new becomes the current program as follows:

① Press INPUT key;

② Inputting the program name which is not in the program list, press INPUT key and the current program content is copied to the new program. The new program is taken the current one.

【Note】

When the input program name exists, the system prompts **E161 program to be copied already exists** and waits the prompt losing to input a new one.

【Example】

Example: copy the current program %000 to %005 as follows:

Press key input: INPUT 5 INPUT. The copy operation is completed.

4.3.3 Part Program Communication

The communication of part programming includes the sending and receiving the part program. The sending mode is: the system outputs to U disc (CNC→USB); the receiving mode is: U disc inputs to the system (USB→CNC)

Press hp2 in the program catalog search window, the system opens the U disc to enter the communication interface.

4.3.3.1 Sending Part Programs(CNC→USB)

USB communication;

1） After entering USB communication mode, the system first checks whether a U disc has been inserted. If no U disc is inserted, the system displays a prompt message box, prompting No USB device inserted . If a USB disc containing a folder named “C001PRO” has been inserted, and there are CNCxxx.TXT files in the folder, the files will be listed on the USB file directory box. If there are no CNCxxx.TXT files in the C001PRO folder, the system prompts: USB device specified directory: C001 PRO has no programs. If the folder named“C001PRO”does not exist in the U disc, the system prompts: No specified directory in USB device: C001 PRO

2） Output the part program stored the system to U disc as follows:

① Insert U disc in the system USB interface;

② The system automatically opens U file catalog(create a file in the U disc: C001PRO，file format CNCxxx.TXT) . The system creates “C001PRO” when there is no “C001PRO” in the U disc root catalog.

③ In program search page, press key to input ： hp2→ Edit ;

④ Select the required programs to send according to the system hp1 help message.

⑤ Press ENTER to send and the system prompts to select the sending mode;

⑥ The system outputs the selected programs based on the selected sending mode and displays the sending process till the sending is completed;

⑦ Select ESC to exit from U disc.

4.3.3.2 Receiving Part Programs (USB→CNC)

USB communication;

1） After entering USB communication mode, the system firstly checks U disc, and opens "C001PRO" existed in the U disc and lists "CNCxxx.TXT" file.

2） Output part program stored in U disc to the CNC system as follows:

① Insert U disc in the system USB interface;

② In the program list index page, press key to input：hp2; the system will automatically open U disc file catalog;

③ Select the required programs to receive according to the system hp1 help message;

④ Press ENTER to receive and select the sending mode according to the system prompt;

⑤ Input the programs according the selected receiving mode, display the receiving process till the receiving is completed;

⑥ Select ESC to exit from the U disc.

【Note】

If the stored program name is the same as the sent program name, a prompt will be displayed for you to determine whether to replace the original program or not. If you choose yes, the original program will be replaced by the received one.

4.3.3.3 TXT Part Program Standard Format in U Disc

Use TXT, LST text to edit part program in U disc, but the file name must be compiled based on the standard format required by the system to be sent to the system as follow:

1) Name the file name of the part program to TXT or LST suffix, such as "CNC008.TXT" in PC; it is suggested that the operator use TXT suffix to conveniently execute part programs on the PC.

2) The first line of TXT file content must specify the program number, its format is "% XXX", i.e. percent sign follows the digit 1, 2 or 3, its range is 0~254, and the first line cannot have other content. Its range must be 0~254 because the program number created and stored by the system is 0~254, otherwise, the system cannot receive the programs and prompts the corresponding mistaken message. The program numbers received by the CNC are determined by the following method:

◆ The program number sent by the system USB is digit xxx of CNCxxx.TXT in"C001PRO" file folder in the U disc root catalog.

Note: in USB communication, the character string XXX digit of %XXX" in the first line should be

the same that of xxx digit of CNCxxx.TXT. When they are different, the xxx digit of CNCxxx.TXT is taken as the standard.

3) The blocks start from the second block. The block must meet its format. Each block cannot exceed 250 characters, ends with the ENTER key, otherwise, the system prompts the error: Block too long in received programs.

4) The annotation area has Chinese annotation in the block.

5) Max. TXT file length cannot exceed the program stored space limit of the system.

6) You can refer to the part program file format that the system outputs to the U disc.

Part program communication standard format in U disc:

TXT file format	Explanation
%099 N0000 G50 X100 Z100 ; Sets coordinate system N0010 G00 X20 Z90 ; Rapid positioning G01 X10 Z80 ; Linear cutting …… /N0250 G01 X30 Z20 ; Linear cutting N0260 M05 N0270 G04 D8 …… M20	**1. There must be program name %099 when the system receives programs; the first line must the 3-digit 0~254+.** **2. "N****" are the blocks with the line number, and others are the blocks without the line numbers;** **3. The home of each line must be blank;** **4. there is a space between the line number and the command for the program with the line number;** **5. / block skip;** **6. ; The following is the annotation.**

4.3.4 Part Program Content Input and Edit

Each input part program consists of many blocks, and each block is composed of block number, command and data. The program format must meet the general programming rules, and there is no prompt in program compiling, and the alarm prompts are referred to ***PROGRAMMING*, Chapter 6 Alarm Message**. The qualified parts can be machined only according to the technology requirements and orderly input correct program contents.

The edit mode of the system is full screen. The program edit window is displayed as Fig. 4-3:

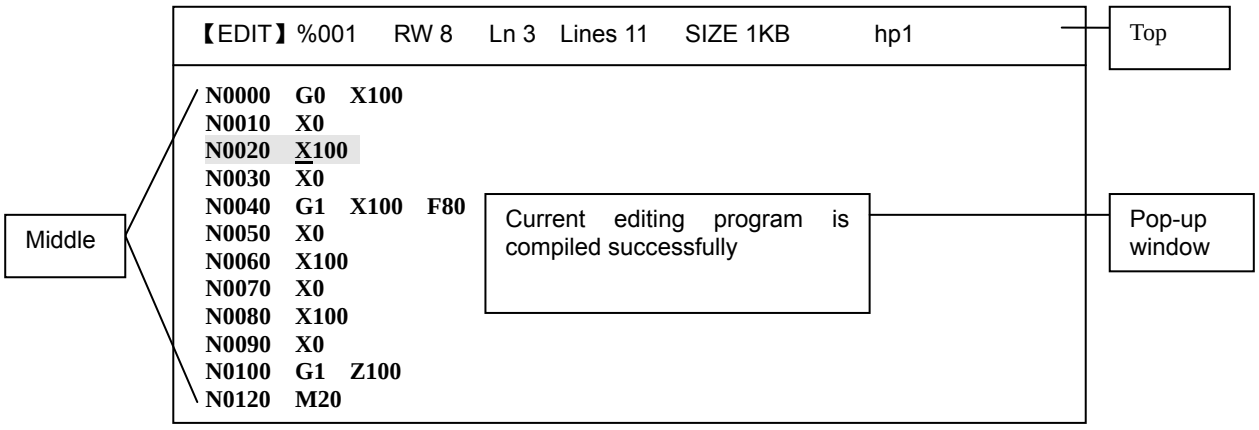


Fig. 4-3 program edit

◆Display content in window area:

Upper top: program number, program capacity(program length) and program line quantity of current program, edit cursor line and row (prompt symbol of current editable character position), and the system function operation prompt key hp1;

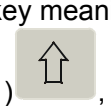
Middle: program edit window;

Pop-up window: display operation prompt message.

【Note】

1. **Press hp1, the system prompts "Program edit help message prompt", introducing all help key explanation.**
2. **When P416_d0 is set to 1, the system forbids pressing key to edit and modify programs, and prompts the alarm E174：machining programs are locked and are forbidden to modify; when the system edits and modifies the programs, P416_d0 should be set to 0.**

Edit key meanings and use in program edit window:

1)  , cursor UP, DOWN move key: Press the move key every time, and the cursor moves up(down) till the top (the bottom) line. Keep pressing the moving key and the cursor continuously moves up (down) till the first (last) line or the move key is releases. In the character string search function (hp1), the operator can search the required character string up and down.

2) , cursor LEFT, RIGHT movement key: Press the move key every time, and the cursor moves left (right) one character. Keep pressing the moving key and the cursor continuously moves left (right) till the first (last) character or the move key is releases.

3)

Home key: the cursor rapidly moves to the home or the first field head of the line. Press continuously the head key, and the cursor switches between the head and the first field of the line. The compound of the head key and the deletion key can delete the current line.

4)

End key: the cursor rapidly moves to the end of the line.

5)

insert/alter: Changing edit input mode: switch the insert and the alter after pressing it once. The cursor also changes correspondingly and the insert mode cursor flashes to be one horizontal line, and the alter mode cursor flashes to be the high light square.

6)

input key: The program edit state is switched into the program catalog search state, and the system prompts Input program number: .

7) Page Up, Page Down: Paging to display the program content. In hp1, the cursor directly moves the home/the end window1.

8)

Enter key: When the cursor is in the first line or the last line, press ENTER, it will locate to the new line.

9)

delete key: Delete all blocks or characters in the block.

10)

hp1 key：Program edit help message prompt.

11)

hp2 key：Current program compiling.

Multi-function definition key input must obey the following rules:

1) When the first letter is capital in the line, the first key value is prior; when it is the lowercase, the 3rd or 2nd key value is prior.

2) The system automatically creates the blank space when the letter or character following the digit (0~9) is input.

3) The cursor stays the position where the input is convenient after the character string is input.

4.3.4.1 Inputting Program Content

Note: P333 is set to 10 (the system automatically creates the block number, and the following is the same).

In program edit window, inputting the part program content is as follows:

① create a new program according to the operations of creating new part program;

② input one line content after the block **N0000** is displayed;

③ press ENTER key after inputting one line programs to end the line input;

④ the system automatically creates the next block number and continuously input the program content;

⑤ press ESC to complete the program content input after the last line programs are input.

【Note】

1） The first row of every line only displays the blank space;

2） Each block only displays 60 characters, and only the first 60 characters are displayed when there are more than 60 characters, you can press to left move one character.

3） The serial number of the first row blank is 1, the last is 252, and the system only displays the cursor instead of the character; there are up to 250 characters in the edit line.

4.3.4.2 Inserting Program Line

Insert one or many program line between two program lines or in the front of the current block as follows:

① Move the cursor to the first block end or the last block home of the two blocks;

② Press ENTER, and the system automatically creates a new block number between the current block and the next (the serial number increment is P333 parameter 1/4 integer, the next block number can be modified when the above the increment is not enough) and remains a blank line.

③ Input the required block content;

④ After all content is input and many line are required to insert, ENTER is pressed, which is not done when one line is required to insert.

【Example】

Example: insert a new block **M3** between **N0020** and **N0030** as follows:

Move the cursor to the end of block **N0020** or the beginning of block **N0030**; Press ENTER key, and input M3.

4.3.4.3 Deleting a Block

Delete all content in one line as follows:

① Move the cursor the home of the required line;

② Press DELETE;

③ Delete all content of the selected line.

4.3.4.4 Inserting a Character in a Block

Insert a character in one block as follows:

① Press ALTER, switch the input mode to insert mode, i.e. the cursor is displayed to the down horizontal line;

② Move the cursor to the character following the required insert position;

③ Input the required insert content;

④ Insert the input content before the character pointed by the cursor.

【Note】

The CNC system requires there is a blank space between fields in the program line. In editing program, the system can automatically judge and create a blank space, but cannot automatically judge in the insert operation, and at the moment, the operator should input a blank to get the complete program.

【Example】

Example: insert **1** between **X** and **0** in **N0020 G0 X0.0 Z0.0**. As follows:

The cursor moves the under **0** following **X** input **1,** displaying **N0020 G0 X10.0 Z0.0**

4.3.4.5 Deleting a Character in a Block

Delete the content, which is not needed as follows:

① Move the cursor to the character position which is needed to delete;

② Press DELETE to delete the character.

4.3.4.6 Modifying a Block Content

Modify the content of the block into the new, which can be complete according the input mode (insert/alter).

In INSERT mode, use the insert and deletion as follows:

① Move the cursor the character which needs to be modified;

② Input the new content;

③ Press DELETE to delete the required content.

In ALTER mode, directly modify the content as follows:

① Press ALTER; the cursor switches into the alter mode(the character position pointed by the cursor is high light square);

② Move the cursor to the character which needs to be modified;

③ Input the new content. The cursor points the next character.

【Example】

Example: Alter **X** in **N0020 G0 X0.0 Z0.0** into **U** as follows:

Switch the input mode into ALTER mode, move the cursor to the down of X and input **U**. Alter to N0020 G0 U 0.0 Z0.0 .

4.3.4.7 Program Stored Space

For programs No. 0～252, No. 254, the system provides 400K memory space, so theoretically, the size of a single program can be up to 400K. For No. 253 program, the system provides 4M FLASH memory space.

【Explanation】

1) The system displays the program leftover space and prompts the leftover stored space size.

2) When the current edit No. 0～252, No.254 program size are more than all stored space (max. 400K), the programs cannot be saved, and the system prompts the storage space has full: Overflow in edit area. Delete the old programs when the leftover space is not enough.

3) Max. edit space of No. 253 program is up to 4M, and is only saved to the system FLASH.

4) In saving programs, the program capacity is big and saving the programs need long time, and the operator needs to wait.

4.3.4.8 No. 253 Program Operation

Because No. 253 program is up to 4M, its solidifying and read are special as follows:

1）Only be saved to a fixed FLASH;

2）Select No. 253 program, press hp1and then 4 to save it to the FLASH in program edit window;

3）Can use USB to complete the communication.

【Note】

1) Program No. 253 cannot be saved automatically, and should be complied and solidified after editing; otherwise, it cannot be saved. However, if the program No. 253 is transmitted from USB successfully, the system will automatically save this program.

2) Program No. 253 cannot be copied or renamed.

3) Longer time may be needed when the saved program No. 253 is large.

4.3.4.9 No. 254 Program Operation

For No. 254 program, press hp1 in program edit window, the system prompts the help message prompt box how to compile, solidify and read No. 254 as follows:

1) Press 4 and read No. 254 program:
Read No. 254 program saved in FLASH area to the edit buffer zone, and update it.

2) Press 5 to compile and solidify No. 254 program:
Edit No.254 program. The system alarms when the edit is wrong; the system saves it to the FLASH area when it is compiled successfully.

3) Press Key 6 to delete the custom command of No. 254 program
If there is a custom command in the solidification area, the command will be displayed on the screen. After the custom command in No. 254 program is deleted, there is no custom command in the system solidification area; the help information dialog box for No. 254 program prompts “No custom command in solidification area”

4) Press ESC to exit the current state.

[Notice]

If the bit parameter P404_d0=0, the part program of No. %254 can’t be solidified, picked and deleted.

4.3.5 hp1 Function

Hp1 help key including the system command help, macro string insertion, line number sort, character string replacing, cursor positioning and so on.

When the current program is No. 253 or No. 254, hp1 help key adds its operation prompt.

The movement of the MPG control cursor is controlled by MPG key.

4.3.5.1 Part program Command Help

In program edit window, press hp1 1, the system prompts “Command help introduction” window; the function can search all commands of the system including G, M, S, T, and F as follows:

1) G, M, S, T, F separately introduces G, M, S, T, F command.

2) Press INPUT and input the command number to search; the system displays the definition, the function, the format and the explanation of the command number.

【Example】

Example: search G01 command help.

Press ： hp1 1 INPUT G 01 ENTRE, the system displays G01 definition, function, format and explanation.

4.3.5.2 Inserting Macro String

The procedure of inserting macro string on the program edit page is as follows:

① Press hp1 2 key to display the macro string list on the screen;

② Select the desired contents according to the keys;

【Example】

E.g. Press hp1 2 key, then press G key, program contents input: **r = r * r / r**.

【Note】

For details about the variables and statement programming explanation, refer to Chapter Seven Programming Statement.

4.3.5.3 Line Number Sort

Press hp1 3 in the program edit window, and the system sort again the programs, and the sorted block number increases in 10 times. (**P333** value is set to 10.**)**

【Note】

1）After sorting the block number again, the program skip error appears when the program skip command is used in programming.

2）Program sorting function is invalid when P333 is set to 0.

4.3.5.4 Replacing Character String

Press hp1 R in the program edit window, and the system prompts “Character string replacing” window; the operator can execute the operation according to the system prompts. All characters which need to be replaced are replaced from the character where the cursor is to the last character.

4.3.5.5 Cursor Position

The system provides the character string search function, i.e. position the content needed by the operator, which is convenient for operator to search the required content. Press hp1, and the system prompts , , , , F operation functions as follows:

1) Press and the cursor positions to the first page of the current program.

2) Press and the cursor positions to the last page of the current program.

3) Press F, input the character which is needed to search of the current program, press ENTER, the cursor positions the character.

4) Press , to search the character of current program according to the system prompt. The system prompts Searching is completed and there is no character string when there is the character which is needed to search of the current program.

4.3.5.6 MPG Controlling Cursor Moving

After the system is connected with MPG, the operator presses MPG, rotates MPG to control the cursor movement when the MPG key LED is light on the operation panel. Press MPG repeatedly, MPG operation is invalid when MPG key LED is OFF. The concrete MPG connection is referred to **CONNECTION**.

4.3.6 Part Program Compiling

The system provides hp2 compiling command key to compile part programs, check the syntax error, logic error of programs and coordinate data overtravel according to part program execution path to reduce the alarm error in Auto mode and improve the safety of executing part programs.

Press compiling command key and the system orderly checks and compiles part programs from the first line block, and creates convenient target command according to the execution path. When the system finds out the mistaken operator programs, it stops the compiling; displaying the mistaken field position and number in the line of the source program, prompting the operator to modify till the mistaken is corrected.

4.3.6.1 hp2 Compiling Command

In program edit window, press hp2 and the system orderly compile the current program. The system pops up a window『Program alarm』when it finds out a mistaken message. The system displays Current edited program compiling is completed successfully when all command compiling are correct.

『Program alarm』message includes as follows:

Error: mistaken command (refer to **PROGRAMMING, Chapter 6 Alarm Message** according to the commands);

Program: content of mistaken block;

Position: mistaken letter or field of mistaken block.

【Explanations】

1) Program which is compiled successfully by hp2 can run in AUTO working mode.

2) The system automatically completes the compile when it switches from EDIT working mode to other working modes.

3) Press hp2 to complete compile to appear『Program alarm』, and then press any keys and the edit cursor automatically points to the mistaken block.

4) Executing hp2 compiling, the system assumes that the machine coordinate axis takes the current workpiece coordinate position as the starting point, starts the execution from the first block of the current program. So, for some special programs, each axis stopping position has effect on the compiling; it is suggested that each coordinate axis should stop in advance in the starting point of the machining.

4.3.6.2 Program Compiling Result Analysis

In program edit window, the program compiling error creates two types of alarm: **［Program alarm］** and **［Program compound check alarm］**. The program compiling is completed successfully there is no the above alarm.

Program alarm: there are mistaken commands in programming to cause the alarm, and the correct command is input to clear the alarm, which is not related to the parameter setting.

Program compound check alarm: program command check causes the alarm, which is resolved by the setting of all auxiliary parameters and interface parameters to analyze the program, and then by modifying program and parameters to clear the alarm.

【Example】

［Program alarm example］:

In compiling program, press hp2 and the alarm display is as follow:

```
［Program alarm］
Error E206：Missing message
Program：N0100 G32 Z300
Position： Z
```

［Program compound check alarm example］:

Press hp2, and the current edit program compiling is completed successfully, when the system is switched from EDIT working mode to AUTO working mode, the alarm display is as follows.

```
［Program compound check alarm］
Error E610：Have illegally used M78
            The command function is invalid
```

4.3.6.3 Program Compound Check Prompt

After the program is compiled, the program is executed in AUTO working mode when there is no error. The system displays the program compound check prompt as follows:

1) Tool nose coordinate software limit, machine coordinate software limit exceeding range

In executing programs, the system displays the program compound check prompt when the tool nose software limit and machine coordinate software limit exceeds the setting range set by the

parameter from EDIT working mode to AUTO working mode.

4.4 JOG Working Mode

In JOG working mode, the operator can directly press the function key to execute some operation, and also press the letter key execute the some setting or execute some operation; the system provides the corresponding prompt message for each operation.

The relative setting or operation input format and example are as follows: the required function key is expressed with icon; the required input letter or digit key is expressed with the underline; the system prompt is expressed with the frame.

Press to clear the mistaken digit and input it again when the mistaken digit is input in the course of inputting letter or digit.

Press

to exit the current operation before the confirmation when the operator sets some operation or executes the input or man-machine dialog process.

In JOG working mode, the system displays in the top right. Press it and the system pops up one window, displaying the operation key catalog in JOG working mode; press it again and the window is closed; directly press other functions and the window automatically closes.

Press

to enter JOG working mode.

For the CNC machine, its electricity part installation debugging, motion performance debugging, coordinate system creation and tool preparation are completed in JOG working mode.

The system should be performed the essential analyzing and pre-inspection combining the user parameter list and cutter compensation value etc. after entering the Manual page, which will close the manual operation function and display the alarm information if the Manual operation may be generated a **severe unexpected results** during pre-inspection. In this case, modify the parameter based upon the alarm information, and then correctly perform the Manual operation.

The system provides many part program execution mode. The operator can execute many necessary settings in JOG working mode before run to get the safe machining process.

◆ Main function in JOG working mode including:

☆ Coordinate axis moves in JOG mode, STEP mode and MPG mode

☆ Coordinate axis moves in absolute movement mode, relative movement mode

☆ Create machine coordinate system, workpiece coordinate system

☆ Spindle, chuck, cooling, tool post and other miscellaneous function operation

☆ Tool setting operation

☆ Machine real-time state display, pop up real-time alarm

JOG window display is as Fig. 4-4:

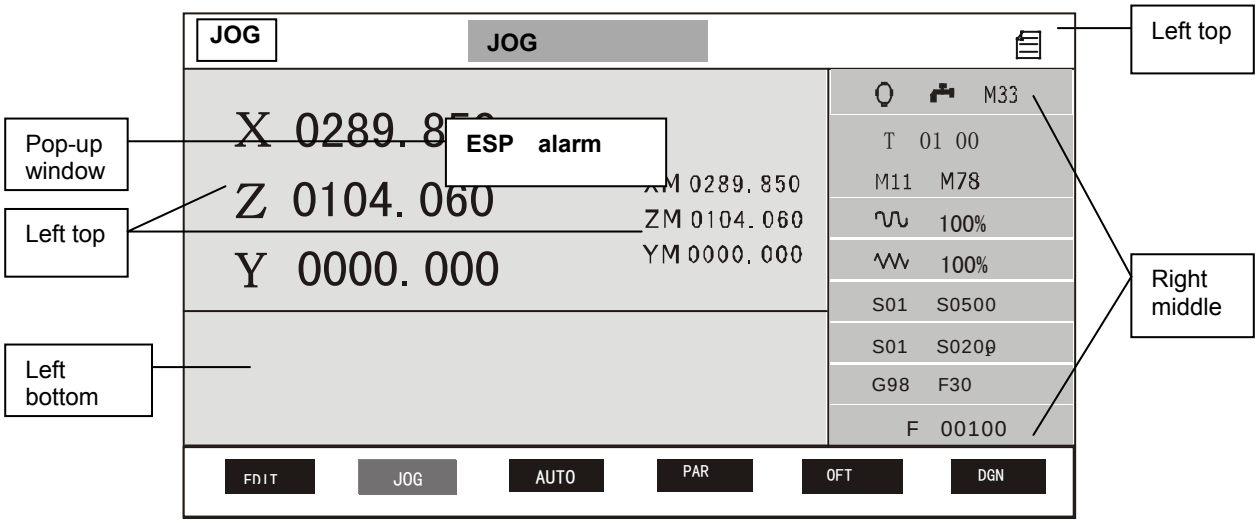


Fig. 4-4 JOG working mode

◆ window area display content:

Upper top: display manual feed operation mode, including JOG, STEP, X/Z/Y MPG; the system function operation way prompt key ;

Left top: display tool nose coordinate and machine coordinate;

Left bottom: MDI input and alarm prompt area;

Right middle: display machine's current state including spindle, cooling, lubricating, tool post, chuck, tailstock, speed, cutting speed and so on;

Pop-up window: display system's alarm message.

◆ Miscellaneous function state display:

1) Miscellaneous function state uses the icon or correspond command symbol display;

2) Black sign indicates the current state: spindle, cooling;

3) Red sign indicates the function is being executed and is not completed;

4) Red flash indicates the last execution is failure or broken in midcourse (reset, emergency stop operation), and the system takes the corresponding function is in the unconfirmed state. When the tool or chuck is in the unconfirmed state, the system cannot start the machining program; the system recovers the normal state when it executes successful operation or is switched on again.

5) The green sign indicates the check is normal and the yellow sign indicates the check is abnormal;

6) The S following the spindle gear indicates the real-time checking spindle speed;

7) The pressure check icon △: it is green when it is normal, it is yellow in half when it is low, yellow in full when the time of low pressure exceeds the time set by P332;

8) G96/G97 and S indicate whether the system is in the constant surface speed cutting mode; S is the linear speed or rotation speed, unit: m/min, r/min;

9) G98/G99 is the feed/rev or feed/minute mode; F is the set cutting speed;

10) F indicates the actual speed of the coordinate axis.

4.4.1 Coordinate Axis Movement

⚠ Confirm the traverse speed and movement distance before executing the coordinate movement. Press the emergency stop button immediately when there is the unexpected accidence.

4.4.1.1 JOG Movement

Press

the system is switched from STEP or MPG mode into JOG mode.

【JOG TRAVERSE】is to keep pressing the coordinate axis move key, and the machine slider continuously moves; release the key, and the slider decelerates to stop. The traverse speed is executed according to the rapid speed or feedrate. However, the max. speed of Z, X, Y is limited by P100, P101, P102.

Coordinate axis move key meanings are as follows:

X/Y/Z negative move key (it is X or Y when

indicator is ON)

X/Y/Z positive move key (it is X or Y when

indicator is ON)

◆ Z/X/Y coordinate axis switch:

Press Z/X/Y SWITCH to the cycle switch of Z, X or Y operation; Z/X/Y SWITCH INDICATOR ON indicates X or Y operation. The selected axis has been shown in the figure above.(when bit parameter **P405_d1**=1 and **P405_d0**=1, the machine tool has Y, X axis, then the switch is valid).

When the machine is switched into X (Y) operation, the program reference point return and the machine zero return are valid in X (Y) axis.

【Note】

1) When the motor rotates with high speed and the feed key is released, the machine slider continuously moves and does not stop immediately because of the automatic acceleration/ deceleration. The movement length is determined by the max. motor speed, the acceleration/ deceleration time, and the feed override. The longer the acceleration/ deceleration time and the higher the speed is, the longer the distance between sliders are; on the contrary, the shorter the distance is.

4.4.1.2 Step Movement

Press

and the system is switched from JOG or MPG to STEP mode.

【STEP TRAVERSE】: Press the coordinate axis move key and the machine slier moves the

preset step width. The traverse speed is executed by the selected rapid or feedrate. However, the max. speed of Z, X, Y is limited by P100, P101, P102.

Press continuously the key, the machine slider will continuously feed the step width till the key is released and the slider has moved the last step width. The step width in the single step movement is displayed in the black background.

The step width of single step movement is **0.001 0.01 0.1 1.0 10.0 50.0. the system can gradually select them according to STEP REGULATION.**

【Note】

1) In STEP mode, press CYCLE PAUSE to stop slider moving. When the key is pressed down, the slider stops and the unfinished step will not be reserved.

4.4.1.3 MPG Control

Press

to switch from MPG mode to JOG or STEP mode, and the indicator ON means it is selected.

【MPG movement】: The system receives the pulse signal generated by the manual pulse generator (handwheel or MPG) to control the movement of coordinate axis. The user can control the slight movement of the coordinate axis by rotating the MPG.

◆ preset each movement of scale of MPG:

The MPG dial rotates one case, and the coordinate axis moves one step width. The step width has three gears: 0.001mm, 0.01mm, 0.1mm, which can be switched circularly according to the STEP REGULATION.

◆ preset MPG coordinate axis:

Press MPG, then the coordinate of the selected coordinate axis is in the high light state. Then press Z/X/Y/ SWITCH key to circularly select X, Y or Z as the axis controlled by MPG;

Rotate MPG after the required coordinate axis is selected, and the selected axis moves according to MPG rotating.

Rotate CW MPG and the coordinate axis moves positively. Rotate CCW MPG and it moves negatively.

【Note】

1) The MPG speed should be less than 5 r/s; otherwise, the motor will not stop even if the MPG stops, that will cause inconsistence in the slider movement distance and MPG scale**.**

2) In MPG mode, all the functions related to the axis moving including JOG or STEP movement function, program zero point return and machine zero point return are invalid.

3) In MPG mode, when axis movement related functions are executed, and the relative/absolute movement of field is input, MPG is forbidden temporarily and the LED flickers. After the above functions are executed, MPG function recovers automatically, and the other S, M, T auxiliary functions are valid.

4) When the bigger override (X 0.1 mm) is selected, the motor will rapidly traverse if the MPG is

rotated rapidly. At the moment, because the system automatically accelerates/ decelerate, the motor will traverse not to stop although the MPG stops. The actual moving distance is determined by max. speed of motor, the acceleration/ deceleration time, the feedrate override and the MPG speed. The rapider the speed is, the longer the acceleration/deceleration time is and the rapider the MPG speed is, the longer the moving distance of motor decelerating is, otherwise the shorter the moving distance of motor is.

5) **P400_d4=0**: the step width 0.1 is valid; **P400_d4=1**, the step width 0.1 is invalid.
6) When **P400_d1** is set to 1, the external MPG control button is valid, Y/Z selection axis, and the step regulation key are invalid.

4.4.1.4 Rapid Traverse Speed Selection

◆ Manual rapid traverse and low feed state selection

In JOG mode, the negative/positive movement speed of each axis can select rapid traverse and cutting feed (low speed movement). Press

to switch the rapid traverse and low speed feed states. The speed indicator ON is to select the rapid traverse state.

◆Rapid override

Rapid override is divided into the four gears: 25%, 50%, 75%, and 100%.

increases to one gear till 100%.

reduces one gear till 25%.

Z (X, Y) actual traverse speed = P100 (P101, P102) x rapid override

◆ Manual operations influenced by rapid override and feed override are as followed:

JOG MOVEMENT operation: when the speed indicator is ON, it is influenced by the rapid override; when it is OFF, it is influenced by the feedrate override;

STEP MOVEMENT operation: when the speed indicator is ON, it is influenced by the rapid override; When it is OFF, it is influenced by the feedrate override;

INPUT FIELD MOVEMENT operation: when the speed indicator is ON, it is influenced by the rapid override; when it is OFF, it is influenced by the feedrate override;

Program reference point return operation: it is influenced by the rapid override; machine zero return operation: it is influenced by the rapid override;

【Note】

1) Firstly select the rapid override and press the coordinate axis movement key in JOG working mode.
2) Select the rapid override in STEP working mode, and regulate the rapid override in the course of movement, and the traverse speed changes.

4.4.1.5 Low Speed Feed Speed Selection

Press RAPID/FEED and the speed indicator is OFF, which is the selected low speed feed state.

◆ System embedding feedrate

When the input field F is 0, the system uses the embedding speed feed.

The manual feedrate override has 16-gear **0%～150%**, the corresponding embedding feedrate of each gear is as follows:

Feedrate override	feedrate(mm/ min)	Feedrate override	feedrate(mm/ min)
0%	0	80%	240
10%	7.5	90%	300
20%	22	100%	420
30%	38	110%	525
40%	60	120%	675
50%	82	130%	850
60%	110	140%	1000
70%	180	150%	1260

【Note】

1) There is an error between actual speed and data in this table, and the actual speed should prevail.

2) When the feedrate override is 0, the system prompts “Feedrate override is 0”, which indicates the command is in execution state, and the machine slider is in stop state. The slider will move as soon as the override is adjusted to a non-zero value.

◆Feedrate override

The manual feedrate override has 16 gears **0%～150%** ; press F.OVERRIDE and the feedrate override increase one gear till 150%; press F.OVERRIDE and the feedrate override reduces one gear till **0%**.

4.4.1.6 Inputting Field Moving, Setting Feedrate

In JOG working mode, the coordinate axis moves according to the input length and direction, or directly moves from the current position to the input coordinate position instead of the set step width as follows:

◆ Relative field of movement

Corresponding operation of each coordinate axis:

X move_X field (X position), or U field (X relative movement);

Y move_Y field (Y position), or V field (Y relative movement);

Z move_Z field (Z position), or W field (Z relative movement);

X, Z, Y, U, W, V range: -99999.999mm～99999.999mm;

Feedrate_F field (F0000～F15000，the leading zero can be omitted, unit: mm/min)

【Move field format】

Z(W)_ F_ ; Z moves, its speed is determined by the rapid/feed state when F is omitted. (The same as in the followings.)

Y(V)_ F_ ; Y feeds

Z(W)_ X(U)_ F_ ; Z/X feed simultaneously

【Note】

1) In JOG working mode, up to 2 axes move simultaneously at the currently selected manual traverse speed.

2) X (U), Y (V) can be input when X, Y set by the system parameter are valid.

【Example 1】

Input: ：W MOVE W –5.2 ENTER RUN? START (or ESC cancel); it means that Z axis moves **5.2mm**along with the negative direction.

Input: ：X MOVE X 40 ENTER RUN? START (or ESC cancel); it means that X axis moves to the place where the coordinate is **40.**

【Example 2】

Input: **Z200 U50** ; Z moves to 200, X moves 50 along with the positive direction. its speed is not specified and determined by RAPID/FEED state.

Input: **U20 W-50 F80**; X positively moves 20, Z moves 50 along with the negative direction. The feedrate is 80 and is influenced by the feedrate override.

Input: **F200**; the set feedrate is 200. The system is switched into feed state.

Input: **F0**; the set feedrate is 0, the system is switched into feed state and uses the embedded speed.

◆Traverse speed explanation:

1) When field F is input, the system will automatically switch to low-speed feeding state and the speed indicator is OFF; the gear of feedrate override is the current gear;when the speed is higher than P113, the speed set by P113 should prevail.

2) When the input F field is 0, the system will adopt the internal speed as the feedrate.

3) When the JOG or STEP is performed, it is under the control of F filed even when the speed indicator is OFF.

4) Under the condition that no F field is input, when the speed indicator is ON, the movement is rapid traverse and affected by the rapid traverse override; when the indicator is OFF, the movement is low-speed traverse and is affected by feedrate override.

5) Under the condition that no F field is input, when the indicator is OFF, the speed of low-speed traverse is limited by P113 (maximum cutting feedrate); when the speed is higher than P113, the speed set by P113 should prevail.

6) Under the condition that no F field is input, when the indicator is OFF, the movement and AUTO mode of two-axis low feed is the same as G01 (interpolation feed). These two axes move simultaneously at a proportional speed and stop at the same time. F is the resultant speed of the two axes.

7) Under the condition that no F field is input, when the indicator is ON, the rapid traverse rate

of single-axis feed is determined by parameters P100~P102 (rapid traverse speed) and rapid traverse override.

8) Under the condition that no F field is input, when the indicator is ON, the rapid traverse rate two-axis rapid traverse is determined by P400_d3, and it is the same to the execution of G00 in AUTO mode. When P400_d3=0, the movement is performed separately. The resultant speed displayed on the screen is larger than the maximum rapid traverse speed of each axis. When P400_d3=1, the movement is in interpolation mode and these two axes move simultaneously at a proportional speed and stop at the same time. The movement parameters and slope of the distance will be taken into consideration so as to ensure that there is no stall during the movement. Different slopes may correspond to different rapid traverse speed.

9) When the feedrate override is 0, and the system is in low-speed feeding mode or F field exists, movement is forbidden. Press ENTER, a prompt "feedrate override is 0" will be displayed till the override is adjusted to other value.

10) The input of F field is specified by G98 command; G99 command cannot be input in the system.

◆ **Call field execution**

The system automatically saves the last 8 times executed command record in inputting field moving.

Press , and the system pops up the window list record; the operator can input according to the digit serial number to modify or directly execute the operation.

4.4.1.7 Drive Unit Enabling Control

Set P416_d4 to 1, continuously press DELETE twice in the overall Manual and AUTO modes, and the drive unit is closed and the motor is in free state. Press DELETE in the drive unit closing state, and the drive unit is started, and the motor is in working mode.

4.4.1.8 Coordinate Axis Motion Alarm Prompt

In executing the axis motion, when the current motion axis meets the tool nose coordinate software limit points (tool nose software limit point), the axis cannot continuously move and moves reversely, and the system displays the alarm prompt. When the axis meets the machine coordinate software limit point, it only moves reversely. But the manual zero return function is not control by the software limit value range.

【Notice】

The system can be prompted the limit alarm information when the motion axis reaches to the soft limit point in the JOG or Single-step mode. When Z/X axis moves inputting command in the key field movement mode, if the specified value exceeds its range; the system prompts limit alarm information instead of executing. Whether the mechanism and cutter soft limit alarm are enabled is set by parameter **P404_d4, P404_d3.**

4.4.2 Creating Coordinate System

4.4.2.1 Creating Machine Coordinate System - Machine Zero Return(Machine Reference Point Return)

◆ Machine zero:

Machine coordinate system fixed on the machine is the reference coordinate system for CNC counting the coordinate position. After the system is installed, the operator should firstly create the machine coordinate system.

The reference point of the machine coordinate system is called machine zero (or machine reference point or machine zero). Some fixed point on each machine is taken as the machine reference point, and the system firstly returns to the machine zero and then returns to the machining starting point to eliminate the machine coordinate system deviation caused by the power-off and step-out; executing the zero return instead of tool setting again after power-off accidentally can find the machine coordinate system and workpiece coordinate system to continuously machine the workpiece.

In most conditions, the system looks for the machine reference point by the deceleration switch and zero switch installed on the machine; or by the one-turn signal of servo motor as the zero signal only with the deceleration switch. The deceleration switch is generally installed near to the Max. travel of positive Z/X//Y coordinate axis.

◆ Machine zero return operations:

Press

, and Z (Y or X) execute the machine zero return; (Z/X/Y switch indicator lighting means Y or X operation)

Z (Y or X) moves to Z（Y or X）machine zero at the selected rapid traverse speed in the zero return direction.

◆ Zero return process as follows:

Zero return mode 1: when there is the deceleration signal and zero signal, the system executes the zero return mode 1; the zero return process is as follow:

Step 1: the coordinate axis moves to the specified direction at the rapid speed till the block presses down the deceleration switch and the system has checked the starting point of the deceleration signal to decelerate to stop moving;

Step 2: the coordinate axis reversely moves at the set zero return speed till the system has checked the starting point of deceleration signal to decelerate and to stop moving;

Step 3: when the set zero offset is not zero, the system continuously moves one zero offset value;

Step 4: the coordinate axis continuously moves at the set zero return speed, and starts checking the zero signal till the system has checked the zero signal to decelerate to stop moving;

Step 5: the above operations have completed the zero return motion and check processes; at last, the system automatically modifies the current machine coordinate into the

"Zero coordinate" set by the parameter.

Zero return mode 2: when there is the only deceleration signal without the zero signal, the system executes the zero return mode 2.

Because there is no zero signal, the system reduces the above the Step 3 and Step 4 compared to the zero return mode 1; the system only executes the above Step 1, Step 2 and Step 5 to complete the zero return process, which zero return precision is worse than the zero return mode 1.

Zero return mode 3: when there is the zero signal without the deceleration signal, the system execute the zero return mode 3.

Because there is no deceleration signal, the system reduces the above the Step 1, Step 2 and Step 3 compared to the zero return mode 1; the system only executes the above Step 4, and Step 5 to complete the zero return process. In the mode, the manual operation moves the coordinate axis to a special position and then the system executes the zero return; otherwise, the result is not correct.

Zero return mode 4: the system executes the zero return mode 4 when there is neither deceleration nor zero signal.

When there is no machine zero check device installed on the machine, the relative parameters are set to 0; at the moment, when the system executes the machine zero return function, it does not check the zero signal and deceleration signal till it returns to the zero coordinate position of the axis.

【Note】

1) The direction of machine zero return points to the zero-return direction, if this direction is positive, then, the axis should in the negative direction.
2) In the machine zero return, the rapid traverse speed of the coordinate axis is controlled by the rapid override.
3) In the machine zero return, the coordinate axis motion is not limited by the software limit parameter.
4) Parameter related to the machine zero return is referred to OPERATION, Section 4.6.4.2.
5) Connection related to the machine zero return and zero return mode are referred to CONNECTION, Section 3.3 Machine Zero Function and Connection.
6) After the system executes the machine zero return, the blue icon of the machine zero return after the corresponding machine coordinates are displayed as the prompt.

4.4.2.2 Creating Machine Coordinate System - Without Machine Zero(Without Machine Reference Point)

(Prompt: the coordinate axis with the zero check device cannot execute the operation.)

The coordinate axis without the zero check device (without deceleration signal and zero signal), can create the machine coordinate system as follows:

【Format】

Input: INPUT **U** NEW COORDINATE VALUE ENTER. The current X machine coordinate is modified into the new coordinate value.

Input: INPUT **V** NEW COORDINATE VALUE ENTER. The current Y machine coordinate is

modified into the new coordinate value.

Input: INPUT **W** NEW COORDINATE VALUE ENTER. The current Z machine coordinate is modified into the new coordinate value.

4.4.2.3 Setting Workpiece Coordinate System

The system uses the floating workpiece coordinate system. The workpiece coordinate system is the reference to tool setting and relative dimension. After the machine coordinate system is confirmed, the workpiece coordinate system should be set.

【Format】

Input: INPUT **X** NEW COORDINATE VALUE ENTER. The current X tool nose coordinate is modified into the new coordinate value.

Input: INPUT **Y** NEW COORDINATE VALUE ENTER. The current Y tool nose coordinate is modified into the new coordinate value.

Input: INPUT **Z** NEW COORDINATE VALUE ENTER. The current Z tool nose coordinate is modified into the new coordinate value.

◆ Actual operation steps of setting workpiece coordinate system are as follows:

Install the trial cutting workpiece on the machine and select any one tool (generally use the first tool in the machining).

1. Setting X workpiece coordinate:

1) Select the proper spindle speed and start the spindle.

2) Traverse the tool to cut a little sidestep on the workpiece, and X does not move.

3) The tool in Z direction moves to the safe position, the spindle stops rotating. The system measures the diameter of cut sidestep.

4) Press INPUT and the system displays SETTING, and press X and the system displays Setting Workpiece Coordinate System X, the operator inputs measured radius value(radius programming); press ENTER, and the system automatically sets X workpiece coordinate.

2. Setting Z workpiece coordinate:

1) Start the spindle, traverse the tool to cut a little sidestep on the workpiece. X does not move.

2) Move the tool to a safe position along with X direction and stop the spindle rotation.

Select a point as a reference point (a fixed point on the machine is better, for example, the chuck surface or other reference plane, so that the new workpiece coordinate system overlaps on the original workpiece coordinate system). Then, measure the distance from the cutting surface to the selected reference point in Z direction.

3) Press INPUT, and the system displays SETTING, press Z and the system displays SETTING WORKPICE COORDINATE SYSTEM Z, input the measured data and press ENTER, the system automatically set Z workpiece coordinate.

Note: The system workpiece coordinate system has been created after the above operations are completed.

【Explanation】

1) Setting the workpiece coordinate system operation only modifies the tool nose coordinates of current point without changing the offset and the machine coordinates. The operation result is that the offset between the workpiece coordinate system and the machine coordinate system is set again.
2) Setting the workpiece coordinate system operation is executed once and is not set later after the system is initialized or the workpiece type is changed (all offset values are cleared to zero).

【Note】

When the actual position of the tool is inconsistent with the one of the workpiece coordinate system due to the step out caused by some special causes, it is inappropriate to use the method of resetting the workpiece coordinate system, because both workpiece coordinate system and machine coordinate system are changed after step-out. In this case, if only the workpiece coordinate system is corrected, without correcting the machine coordinate system, an unexpected "machine coordinate soft limit alarm" may occur.

Proper operations after motor stepping-out as follows:

1) Select the reference point (The tool nose easily reaches and the operator can conveniently observe it) for one couple of tool, measure Z, X coordinates of the point.
2) Move the tool nose to some reference point (the known reference point coordinates);
3) Continuously press the DELETE twice and close the drive unit when the tool nose coordinates are not consistent with the reference point coordinates;
4) Input the field to move and make the tool nose coordinates be consistent with the reference point coordinates (the coordinates change and the actual tool nose does not);
5) Press DELETE to start the drive unit.
So, the machine coordinate system and the workpiece coordinate system are corrected simultaneously.

4.4.2.4 Setting Program Reference Point

In the machine coordinate system, the operator should confirm one position where the tool change can be executed safely when the tool post stops here, and where the workpiece is installed conveniently. The program reference point can be set when the tool post stops at the position which is called the program reference point (**Program zero**). The program reference point coordinates are relative to the machine coordinate system.

【Format】

Press INPUT and the system displays SETTING, press **0** and the system displays Setting program reference point?, at the moment, press ENTER, the system confirms Z/X/Y to be the program reference point.

When the operator sets again the workpiece coordinates after setting the program reference point, the previous reference point coordinates do not change in the new workpiece coordinate system, at the moment, the operator should set again the program reference point. The initial value of the program reference point is X=150 Z=200.

After the operator sets the program reference point, the program reference point return

command G26 and the program zero return operation by the system panel return to the point no matter what the machine slide stops anywhere.

4.4.2.5 Program Reference Point Return

Must confirm the program reference point position before the operator executes the program reference point return, otherwise, the unexpected result brings.

In JOG working mode, the operator directly press the function key to execute the operation. After the key is pressed, the corresponding coordinate axis rapidly returns to the program reference point. When Z/X/Y axis conversion indicator lights up that means X or Y is being executed.

By pressing

key, Z (Y or X) rapidly returns the Z (Y or X) program reference point from the current point. (The Z/X/Y axis conversion indicator lights up that means X or Y is being executed).

【Note】

1) Generally, each axis should stop at the program reference point in waiting for the machining.

2) After the system executes the program reference point return, the green icon of the program reference point return before the corresponding machine coordinates are displayed as the prompt.

4.4.2.6 Recovering the Workpiece Coordinate System and Program Reference Point

In JOG working mode, the workpiece coordinate system and the program reference point have been set. In AUTO working mode, when all executed blocks include G50, the workpiece coordinate system and program reference point have been changed. The operator can use the following operations to recover the workpiece coordinate system and program reference point set in JOG working mode.

【Format】

Input: **G 5 1 ENTER**. Recover the workpiece coordinate system and program reference point set in JOG working mode.

4.4.3 Spindle Control Function

4.4.3.1 Spindle Starting/Stopping Control

◆ Spindle starting/stopping as follows:

In JOG working mode, the operator can directly operate the function keys on the panel or input M03/ M04/ M05 to control the spindle rotation (CW/CCW) and stop). (Feed/spindle hold is invalid in JOG working mode).

Press

or input M 3 ENTER; the spindle rotates CW. The system displays the spindle state and LED lights.

Key M 4 ENTER; Spindle rotates along with the negative direction.

Press or input M 5 ENTER; the spindle stops rotation.

◆ Spindle JOG control

The spindle stop key can switch the spindle JOG control state.

In the spindle stop state, press and the Spindle state icon is displayed in the highlight on the screen and the system is switched to the spindle JOG control state. Press again; the system is switched to the normal-regular state. In the spindle manual state, press , the spindle rotates at the specified speed in the specified time and then stops. (When the specified time is too long, the operator can press to stop the rotation). In the spindle manual state, MDI inputting the spindle control command M03, M04, M05 are invalid. The spindle manual speed is specified by P309; the manual rotation time is specified by P308; and the spindle stops and LED indicator is OFF when the manual time ends. When parameter P308 is 0, the spindle JOG function is disabled.

◆ Interlock between the spindle starting/stopping and chuck:

P402_d5=0: interlock relationship between the hydraulic chuck control and the spindle control

1) When the chuck is released, the system forbids starting the spindle; otherwise, the system alarms "Chuck released, do not open the spindle".

2) When the spindle rotates, the system forbids the chuck operations, otherwise, the system alarms "Spindle does not stop, do not operate the chuck".

◆ Interlock between the spindle starting/stopping and tailstock:

P402_d3=0: Interlock between the tailstock control and the spindle control:

The system forbids operating the tailstock when the spindle is rotating; otherwise, the system alarms "The spindle does not stop and the system forbids operating the tailstock".

◆ Spindle starting/stopping execution process and signal output time sequence:

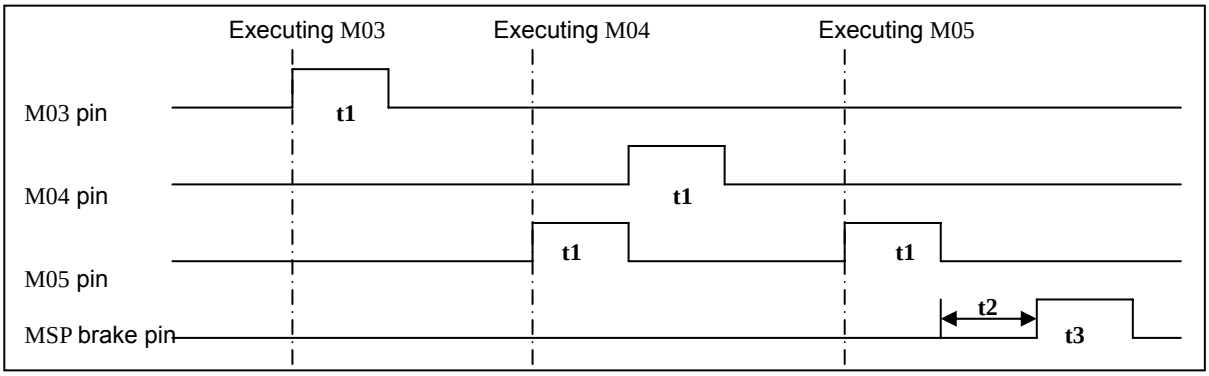
Note: Select the spindle control output signal by P410_d7. When P410_d7 is set to 0, the spindle controls the level output. When P410_d7 is set to 1, the spindle controls the pulse output. The time sequence between the spindle brake signal MSP and the spindle starting, stopping signal as follows:

Wherein: t1: The hold time of M3, M4, M5 signal output is determined by No. P326 parameter in the pulse control method;

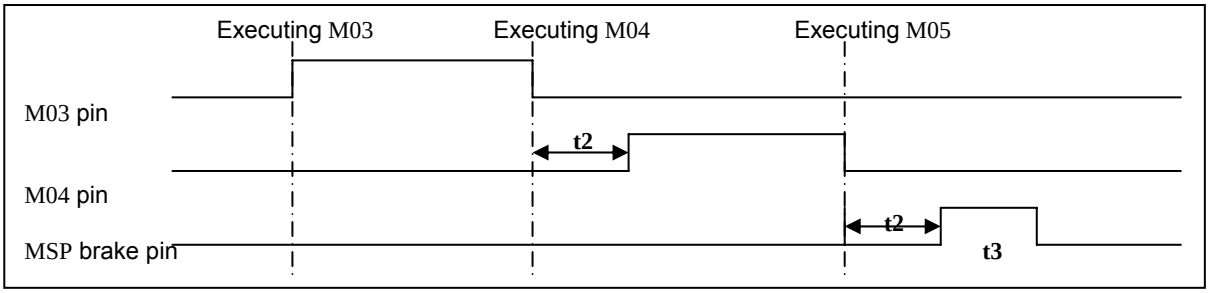
t2: The spindle stopping brake delay time is specified by No. P315 parameter.

t3: The consecutive time output from the spindle brake signal MSP is set by No. P316 parameter.

1） In pulse control mode, M3, M4, M5, MSP output time sequence:



2) In level control mode, M3, M4, M5, MSP output time sequence (it is used to other when M5 pin does not output):



4.4.3.2 Spindle S Command _Gear Shifting Control

(Prompt: it is not necessary to read the chapter for the operator using the frequency spindle.)

When the spindle does not use the frequency spindle, **P410_d6** is set to 0 and S function executes the spindle gear shifting. S standard format consists of S+2-digit number. 2-digit number means the spindle gear number.

【S format 】

Sx ;

Sxx ;

【Operation example】

Select No. 2 gear spindle speed:

Input: S 0 2 ENTER ; the system outputs S02 and the system displays the gear state S02 .

【Explanation】

1) Lines controlled by the Actual output of spindle gear is specified by P310.

P310=4，actual output controlling points are S01, S02, S03, S04;

P310=3，actual output controlling points are S01, S02, S03; releasing S04, as other use;

P310=2，actual output controlling points are S01, S02; releasing S04, S03, as other use;

P310=1，actual output controlling points is S01, releasing S04, S03, S02, as other use;

P310=0，S does not output; releasing S04, S03, S02 , S01 as other use.

2) When the gear controls the signal code output (**P410_d5** is set to 1) and the controlled lines specified by P310 are less than 4, only the low gear control is valid, and the high code control is released and is not controlled by the gear.

3) When **P410_d5** is set to 0, the gear control signal directly outputs by the bit, S range is S00～S04. One gear signal corresponds to one gear signal. S0 means all output is invalid.

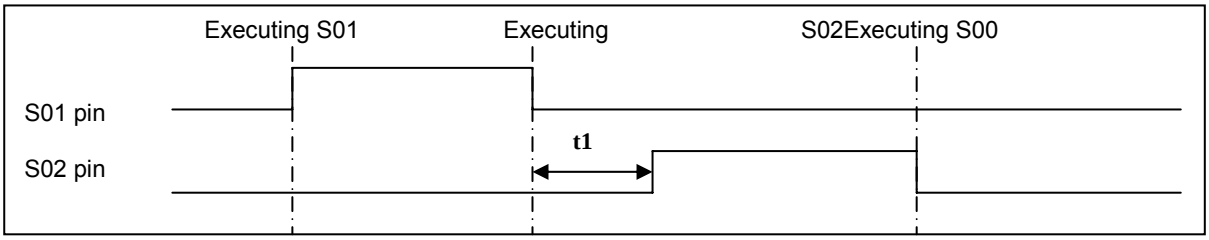
4) When **P410_d5** is set to 1,the gear control signal outputs in code, S range isS00～S15. The detailed code output is as follows:

Code / Output point	S00	S01	S02	S03	S04	S05	S06	S07	S08	S09	S10	S11	S12	S13	S14	S15
S01		★		★		★		★		★		★		★		★
S02			★	★			★	★			★	★			★	★
S03					★	★	★	★					★	★	★	★
S04									★	★	★	★	★	★	★	★

Note: "★" in the above table means the output of the corresponding output point is valid.

◆ Execution process and signal output time sequence of spindle S gear shifting:

When the system is turned on, it defaults S00, S01～S04 output are invalid. When the system executes any one of S01, S02, S03, S04, the corresponding S signal output is valid and keeps, and at the same time, the output of other 3 signals is cancelled. When the system executes S00, it cancels S01～S04 output and one of S01～S04 is valid.



t1：Spindle gear switch interval time (P313).

4.4.3.3 Spindle S_ Speed Control

(Prompt: It is not necessary to read the section for the user who does not use a frequency-conversion spindle.)

When the machine uses the frequency spindle, **P410_d6** is set to 1. To resolve the converter with low torque, the system should 4-gear automatic gear shifting output signal to match the converter working in the high frequency to make the machine get the low speed and big cutting torque. The system uses M41/M42/M43/M44 to control the spindle gear control; S controls the spindle speed.

◆ Frequency spindle gear control

【Format】

M41;

M42;

M43;

M44;

【Explanation】

1) M41, M42, M43, M44 output gear control signal. Each gear signal corresponds to one output point S01, S02, S03, and S04.

2) Actual output controlled lines of spindle gear are specified by P310.

P310=4: actual output controlled point are S01, S02, S03, S04 ;

P310=3: actual output controlled point are S01, S02, S03; the system releases S04 as other use.

P310=2: actual output controlled point are S01, S02,; the system releases S04, S03 as other use.

P310=1: actual output controlled point are S01; the system releases S04, S03, S02 as other use.

P310=0: actual all point doe not output; the system releases S04, S03, S02 , S01 as other use.

3) The initial gear state of the system ON is M41.

[Execution process and signal output time sequence of spindle M gear shifting:]

When the CNC is turn on, it states is controlled by **P400_d6** (spindle gear memory) to whether it memories the spindle gear or not.

1) When **P400_d6** is 0, the system is turned on after it is turned off, the spindle gear is not memorized and the system default the 1st gear of the spindle, and M41～M44 do not output;

2) When is **P400_d6 is 1**, the system is turned on after it is turned off, the spindle gear is memorized.

The system does not execute the gear shifting when the specified gear is consistent with the current gear. If not, the system executes the gear shifting as follows:

① Execute one of M41, M42, M43, M44, the value (unit: millivolt) set by P314 (output voltage in the spindle gear shifting) is output to the analog voltage to the spindle servo or the converter.

② The system closes the previous gear output signal after it delays P311 (frequency spindle gear shifting time 1);

③ The system outputs the new gear signal after it delays P313 (frequency spindle gear shifting interval time);

④ When the system is connected with the checking gear shifting in-position input signal M41I, M42I, M43I, M44I, and the gear shifting is not in-position, it always waits the gear shifting in-position signal to execute the next step; when the system is not connected with the checking gear shifting in-position input signal and directly executes the next step; M41I～M44I input signals are defined by the interface parameters;

⑤ The system delays P312 (frequency spindle gear shifting time 2), and outputs the spindle analog voltage according to the current gear based on **P300～P303** (corresponding to gear 1～4), and the gear shifting ends.

◆ Speed control of frequency spindle

When the machine uses the frequency spindle, S controls the speed. The spindle standard format consists of S+4-digit digital, 2-digit means the spindle gear number. There are 2 methods to

input the spindle speed input.

1) S sets the fixed speed of the spindle (r/min); when S is not changed, the spindle speed is not changed, which is called **Constant speed control**.

2) S sets the tangent speed (m/min) of the tool relative to the workpiece outer, which is called **constant surface speed control**. In the constant surface control, the spindle speed changes as X tool nose coordinate value changing in cutting feed. The detailed is referred to **PROGRAMMING *Constant Surface Control G96, Constant Surface Speed Control Cancel G97*.**

【Command format】

G96		;	set the constant surface speed cutting state;
G96	S__	;	set the constant surface speed cutting state and specify the surface value; range: 0～9999 m /min;
G97		;	cancel the constant surface state; G97 is modal
G97	S__	;	cancel the constant surface state and specify the speed value; range: 0～9999 r /min;
S__		;	It is determined by the current state; it can be speed value or surface speed value cutting.

【Operation example】

Input: S 2 0 0 ENTER; the system switches the speed into **0～10V** analog voltage to output the converter.

【Explanation】

1) In executing S, the system takes the max. spindle speed value of the current spindle gear as the reference, counts the analog voltage value corresponding to the specified speed, and then outputs to the spindle servo or the converter.

2) To make the spindle actual speed be consistent with the speed set by S, **P300～P303** should set the actual max. spindle speed value(output analog voltage is 10V) of each gear; the setting method: input S_ according to the setting value of **P300～P303**, and modify **P300～P303** setting according to the actual displayed spindle speed value.

3) When the system is turned on, the analogy voltage output is 0v, the system outputs corresponding analog voltage value after it executes S; it always keeps later (except for the cutting feed state in the constant surface speed control and X coordinate value is not changed). After S0 is executed, the analog voltage output is 0V. CNC resets in the emergency stop, the analog voltage output keeps.

4.4.3.4 Servo Spindle Working State Setting

The system switches the spindle working mode when the spindle uses the GSK DAP03 servo drive unit. The function should be used if the rotary control and the spindle rotation feeding control (Y axis) are executed for the servo spindle.

【Relative parameters】

The function is valid when the controllable axis has Y.

P410_d4 (relationship between the spindle and Y): it is not switched when it is 0, which means the spindle operation is not related to Y; it is switched when it is 1, which means the spindle operation is related to Y operation, and is interlock but they cannot be operated simultaneously.

When the spindle is switched into the position control mode (**P410_d4=1**) , the spindle speed is controlled by Y feedrate, S is ignored in machining in AUTO working mode, executing S prompts the execution is mistaken in JOG mode.

M47/M48 is valid when **P410_d4** is set to 1.

P410_d1: When M47 is executed, YM is cleared, 0—clear 1—not clear

When P410_d1=0, YM of Y axis machine coordinate will be cleared while M47 is executed.

When P410_d1=1, YM of Y axis machine coordinate will NOT be cleared while M47 is executed.

◆working mode setting operations between the spindle and Y:

In JOG mode, the operator can input M47/M48 to set the working state between the spindle and Y.

Input: M 47 ENTER ; set Y permissive working mode.

Input: M 48 ENTER ; set Y forbidden working mode.

When the system executes M47, it outputs APO level signal“0” and checks API signal; when API level is “0”, the system set Y working mode, displays Y operation icon Ⓨ; in the state, the system permits Y motion operations, forbids the spindle start/stop (M03/M04/05 is invalid and the system prompts the alarm message).

When the system executes M47H1, it outputs APO level signal “0”, not detect API signal and displays the red icon Ⓨ, the system can executed the other commands except for Y movement command, and then executes M47 and checks.

When the system executes M48, it outputs APO signal“1” and checks API signal; when API level is “1”, the system set spindle working mode, Ⓨ disappears; in the state, the system forbids Y motion operations, permits the spindle start/stop (Y motion in AUTO working mode causes the alarm). The concrete connections of APO and API signals are referred to **CONNECTION**.

[Explanations for M47]

1) After the system executes M47, press STOP EMERGENCE, the system will exist M47 status, and it is no use to press RESET.
2) When the system is executing M47, press RESET, or STOP EMERGENCE; The system will exist M47 status.

4.4.3.5 Spindle Orientation Control

M87/M88 is used to control the spindle orientation. The main function of M87/M88 command is:

1) If the spindle transducer is with the function of the spindle orientation in any angle, and the

orientation command is sent into the transducer, and then the orientation signal is output with M87 command through RS232 interface based on Modbus communication treaty, so the spindle orientation in any angle can be realized.

2) The orientation signal is output with M87 command to realize single point or multi-point orientation of DAP03 series spindle servo drive.

[Relative Parameters and Remark]

Miscellaneous parameters: **P342**：M87 communication delay (ms).

P342 is the delay from the communication completed to output YO2 control signal.

Miscellaneous parameters: **P343:** M87 communication address (decimal).

P343 is the system sends the orientation angle Q into the address of the spindle transducer during executing M87.

For example, the communication address of the transducer is H1201(hexadecimal) , and it is switched into the decimal:（1×4096）+（2×256）+（0×16）+1＝4609, so P343 should be set as 4609.

When **P343** is 0, M87/M88 function is invalid, M47/M48 function is valid; the system allows to use M47/M48 command, M87/M88 command can't be used. When **P343** is 99999, skip the step of communication during executing M87 (refer to the M87 command executing process as below). When **P343** is not 0 and 99999 (the range of **P343** is 0～99999), M87/M88 function is valid, M47/M48 function is invalid; the system allows to use M87/M88 command, M47/M48 command can't be used.

Bit parameters: The system communication baud rate is set by **P414_d7 and P414_d6,** 9600/19200/38400 is optional.

[Code Format and Remark for M87/M88]

M87 Q_;The orientation is executed and the system detects whether the orientation is completed or not; the next one can be executed if it is completed.

M87 Q_ H1 ;The orientation is executed but the system does not detect whether the orientation is completed; the system can execute other commands and then detect.

M87;Detect whether the orientation is completed or not; the next one is executed if it is completed.

M88 ;Cancel the orientation signal and retract from the orientation state.

[Explanations]

1）M87 Q means the combination of two commands M87 Q H1 and M87.

2）The range of Q is （-360.000～+360.000）, 0 is positive and 360.000 equals to 0.

3）The code before Q is just represented the rotation direction during the spindle orientation, and the index is calculated based on the absolute value of Q.

4）The variable of r1~r199 is allowed to use in Q field.

[M87/M88 Command Execution Process]

The executing process of M87 Q or M87 Q H1 is as below:

(1) Based on ModBus communication treaty, the message is sent and delay for the time set by **P342;** (The step is skipped if **P343** is 99999 or the message is sent which is same as that of Q numerical value); if the communication is not successful, sending is operated, again.

(2) The orientation enabling signal YO2 is sent, and then delay for 10ms.

(3) If the spindle is in M05 state, output based on Q code: When Q≥0, M03 signal is output; when Q＜0, M04 signal is output; (If the spindle does not with CCW signal M04, CW signal M03 is output no matter whether the direction is positive and negative).

If the previous spindle is being rotated, hold M03/M04 output signal, and orientate along with the rotation direction.

(4) If M87 command is with H1, the command execution is completed; otherwise, the next step is executed.

(5) Delay for 10ms, the orientation finish signal YI2 is detected.

(6) Command execution is completed.

Execution process of M87: (Without parameter for M87)

(1) Detect the orientation finish signal YI2;

(2) Command executing is completed.

Executing process of M88:

Exit the state if it is in M87 orientation mode:

(1) Cancel M03/M04 signal and delay for 6ms;

(2) Cancel the orientation enabling signal YO2 and delay for 6ms.

The above steps are skipped if it is in M87 orientation mode.

[Explanations]

1）After executing M87, the orientation enable signal YO2 and the spindle CW/CCW signal M03/M04 still hold, so CCW signal will be automatically switched off when M87 output CW rotation is executed continuously.

2）In the orientation state, no matter whether the current spindle is rotated or not the spindle CW/CCW indicator lamp on the panel will be ON based on the output CW/CCW signal.

3）After executing M87, the spindle is orientation mode, the spindle is forbidden to switch on; Press emergency stop button, the system exits the orientation mode, and it does not after pressing the resetting key.

4）During executing M87, the system will exit M87 orientation mode if the resetting or the emergency stop key is pressed.

5）During communication, if the communication is overtime or the received Modbus is with the normal information, cycle and try it again after some time.

6）In M88 state, the detection command “M87” is invalid.

[Alarm and information during executing M87/M88 command]

Two following alarms occur during executing M87/M88 command:

1）E383：The spindle is forbidden to operation in M87 state;

2）E384：M87/M88 function can’t be used.

The following information occurs during executing M87/M88 command:

1) “Normal connection: After sending the signal, the receiving responds abnormally; the reasons are the parameter **P343** setting is wrong or the abnormal response caused by the other interference.

2) "Not connected, no response": After sending the signal, the response signal is not received; it may result from the line fault or the incorrect communication baud rate.

3) "Connecting, again……": The system tries to send, again.

[Spindle State Prompt]

1) M03/M04 state is displayed in red: It means the spindle orientation command is executing (During executing M87 Q H1command, the actual orientation may be completed.)

2) M05 state is display in blue: It means the current is in the orientation state, and the spindle orientation has already completed.

3) The spindle is displayed in the normal color: It means it has already exited the spindle orientation state.

Note: 96 system does not lead to RS232 interface; it should be taken out by the technician with the special tool.

4.4.4 Cooling Control

In JOG working mode, directly operate the function on the panel or input M08/M09 to control the cooling ON/OFF.

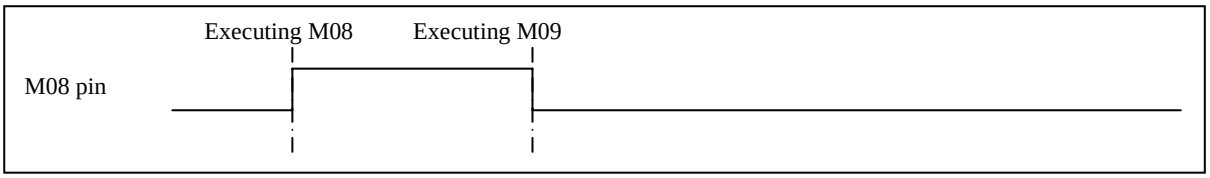
Press

to switch cooling ON/OFF; the State icon on the screen and the LED indicator indicate its corresponding state.

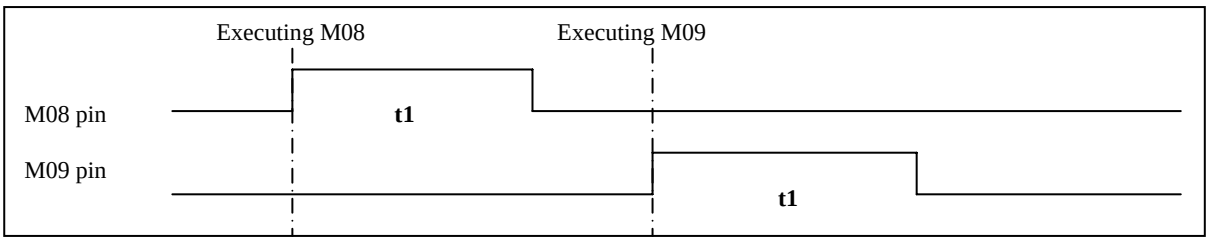
Input: M 8 ENTER ; coolant ON.

Input: M 9 ENTER ; coolant OFF.

1） In level control mode, M8, M9 output time sequence: (**It is used to others when M9 does not output**)



2) In pulse control mode, M8, M9 output time sequence:



Wherein: t1：In pulse control mode, M08, M09 output hold time is set by P326.

【Relative parameter】

P410_d7: **P410_d7=1:** the system pulse output controls the cooling; **P410_d7=0:** the system level output controls the cooling. The bit parameter shares with the spindle controlling output bit parameter.

4.4.5 Manual Tool Change Control

The system has up to 16 tool positions. It selects the tool change control process by setting the tool post type parameter (the system has no the integrated function of the electric tool post on the machine). The tool post type parameter P318 is 0～8, and the machine is equipped with the line-up tool post; When the tool post parameter P318 is set to 0, the tool change operation is not performed; when sets to 9, the system calls M60 command to execute the tool change.

◆ Input format of T command

The standard format of the tool function field consists of T+4-digit, the first 2-digit is the tool number, and the second 2-digit is the tool offset number. It is not necessary to input the complete 4-digit or to use the 2~4-digit.

【Format】

Txx ____ the first 1-digit is the tool number, the second 1-digit is the tool offset number;

Txxx ____ the first 1-digit is the tool number, the latter 2-digit is the tool offset number;

Txxxx ____ the first 2-digit is the tool number and the second 2-digit is the tool offset number.

【Explanation】

Tool number range is decided by P319 (max. tool number: 1～16; when P319 is 4, the tool number range is 0～4.

The input tool number is 0, which means the system keeps the current tool number.

The offset number range: 0～64; the input tool offset number is 0, which means the system cancels the offset.

【Example】

Key in： T 4 6 Enter ； Changing to tool No. 4, and executing tool offset No. 6

Key in： T 0 6 Enter ； Maintaining the current tool No, and executing tool offset No. 6

Key in： T 0 0 Enter ； No tool changing, and canceling tool offset

Key in： T 0 4 0 Enter ； Maintaining the current tool No., and executing tool offset No. 40

Key in： T 4 0 5 Enter ； Changing to tool No. 4, and executing tool offset No. 5

Key in： T 0 6 0 8 Enter ； Changing to tool No. 6, and executing tool offset No. 8

【Note】

1) Example: inputting T400 means the system executes No. 4 tool change and cancels the offset. (Note: cannot input T040).

2) Set P318 to 0, and there is no tool change signal output in selecting the line-up tool post.

3) When the tool change is failure or is interrupted in the tool change (reset, emergency stop), the system confirms the tool is in the unconfirmed position and prompts the tool number flashing in red, at the moment, the operator cannot start the machining program; the system can recover the normal state when it executes one successful tool change operation and it is turned on again.

4) For the tool change mode of M60 self-defined command, refer to Section 3.4 Tool Change

Control Function and Connection in this manual.

4.4.6 Manual Tool Setting Operation

Machining one workpiece needs several different tools. Because of the tool installation and tool shape deviation, its tool nose position is not complete consistent and has some offset when each tool rotates to the cutting position,

Tool setting is called that the system automatically memorizes the Offset to the specified tool Offset number.

After the tool setting, the operator is only based on the part drawing and machining technology to compile the workpiece program without considering the tool deviation, and only specifies the corresponding Offset number in the tool change in machining program.

The offset table can record 64 groups of tool offset, each tool offset number corresponds to one group from 1～64. Each group separately records Z offset, X offset, Y offset (Refer to OFFSET working mode when there is X or Y.)

Through tool setting, the system will modify Z offset, X offset, Y offset of the specified tool offset number.

【Notes before tool setting】

1) According to the above, confirm the Offset to the Offset number, and confirm the content of Imaginary tool nose number in advance.
2) For the same tool nose, memorize Z offset value and X offset value to the same one offset number, otherwise, it causes the serious result.
3) Generally, it is better to use the sequence for No. 1 Offset number with No. 1 tool, No. 2 Offset number with No. 2 tool, which is convenient to memorize them.
4) Firstly execute the Offset number, and then tool setting. Example, firstly execute the T49 command when the system memorizes the Offset in No. 4 tool to No. 9 Offset number.
5) The system executes the tool setting when the workpiece coordinate system is normal, otherwise, the result is not correct.

The system has the trial cutting tool setting as follows:

◆ **Trial tool setting method(method 1):**

【Format】

Input: **k** MEASURED VALUE ENTER [TOOL OFFSET NUMBER] ENTER. Modify the current Z tool nose coordinate into the new one. en start the sp

【Actual operation steps of tool setting as follows:】

When the system controls multiple axes, press

key to select the desired axis for tool setting.

Install the trial cutting workpiece on the machine; execute the tool-setting operation of each tool

through the above process till the tool setting of all tools are performed. The operation is fast and convenient when a tool is regulated.

1. X tool setting:

1) Select the desired X axis to be tool-setting by

2) Install the trial workpiece reliably on the machine, and select a tool (usually select the first one used in machining).

3) Select a suitable spindle speed, and then start the spindle. In the manual method, move the tool to cut a small stage on the workpiece along with the surface.

4) X does not move but Z does to the safe position, and stops the spindle. Measure the diameter of the cut sidestep.

5) Press **K**, and the system displays **Tool setting X** ; input the measured diameter vale and press ENTER.

6)The system prompts **Input the tool offset number：XX** ; it automatically presets one offset number, and the operator directly presses ENTER when the offset number is consistent with the input. Otherwise the operator presses ENTER after inputting offset number. The system automatically counts X tool offset value and stores it to the specified offset number.

2. Z tool setting:

1) Select the desired Z axis to be tool-setting by

2) Start the spindle again, traverse the tool to cut a small sidestep of the workpiece.

3) Z does not move but X does to the safe position, and stop the spindle.

4) Select one point as the reference point, measure Z distance from the cut end face to the selected reference point.

5) Press K and the system display Tool setting Z to input the measured data, and press ENTER.

6) The system prompts **Input the tool offset number：XX**; it automatically presets one offset number, and the operator directly presses ENTER when the offset number is consistent with the input. Otherwise the operator presses ENTER after inputting offset number. The system automatically counts X tool offset value and stores it to the specified offset number.

◆ Trial cutting tool setting method (method 2):

【Operation steps as follows:】

1. X tool setting:

1) Select the desired X axis to be tool-setting by

2) Install the trial workpiece reliably on the machine, and select a tool (usually select the first one used in machining).

3) Select the proper spindle speed, and then start the spindle. Traverse the tool in JOG

Working mode, and cut a small sidestep of the workpiece.

4)X does not move, the operator presses and the system automatically memorizes the tool nose position, and displays the tool setting icon flashing; and then X moves out the safe position and the spindle stops rotating. Measure the diameter of the cut sidestep.

5) Press K and the system displays Tool setting X to input the measured radius value (radius programming), and press ENTER

6)The system prompts **Input the tool offset number：XX**; it automatically presets one offset number, and the operator directly presses ENTER when the offset number is consistent with the input. Otherwise the operator presses ENTER after inputting offset number. The system automatically counts X tool offset value and stores it to the specified offset number; the system automatically cancels the tool setting icon.

2. Z tool setting:

1) Select the desired Z axis to be tool-setting by

2) Start the spindle again; traverse the tool to cut a end face of the workpiece.

3) Z does not move, the operator presses and the system automatically memorizes the tool nose position, and displays the tool setting icon flashing; and then Z moves out the safe position and the spindle stops rotating.

4) Select one point as the reference point, measure Z distance from the cut end face to the selected reference point.

5) Press K and the system display Tool setting Z to input the measured data, and press ENTER. The system prompts **Input the tool offset number：XX**; it automatically presets one offset number, and the operator directly presses ENTER when the offset number is consistent with the input. Otherwise the operator presses ENTER after inputting offset number. The system automatically counts X tool offset value and stores it to the specified offset number; the system automatically cancels the tool setting icon.

[Explanations]

1) In tool setting icon flashing, the operator can execute the spindle start/stop, the coordinate moving; in tool change, the system automatically cancels the tool setting icon and does not memorize the previous tool setting point.

2) Without the tool setting icon appearing, the operator directly presses **K**, and the system takes the current point as the tool setting point.

3) When Y axis is valid, refer to the Z tool setting steps for the Y trail-cut tool setting mode.

[Note]

1) When the system uses the optical tool setting instrument, it must not start the spindle to place

the tool setting point to the intersection of tool setting instrument, other operations are the same.

2) The system automatically creates the tool offset which can be displayed and modified in OFFSET working mode. Refer to OFFSET working mode.

3) When the system uses the line-up tool setting, if the tool is placed at another side of the workpiece, and the measured value along X direction during trial cutting and tool-setting is the negative.

4.4.7 Hydraulic Chuck Control Function

◆ Chuck operation

In JOG working mode, input M10/M11 to control the chuck clamping/releasing.

Input: M 1 0 ENTER ; chuck clamps. The system displays the spindle state.

Input: M 1 1 ENTER ; chuck releases.

Input: M 1 2 ENTER ; cancel the chuck control signal. (use M12 for special chuck device).

[Relative parameters]

P409_d7= 0: the system has the hydraulic chuck control function.

P402_d5=0: interlock between the hydraulic chuck control and the spindle control.

P402_d4=0: the consecutive check of the chuck respond signal is close;

P402_d4=1: the consecutive check of the chuck respond signal is open

P409_d6=0: the hydraulic chuck is outer;

P409_d6=1: the hydraulic chuck is inner.

P409_d5=1: the hydraulic chuck needs the respond check; it is green when the respond signal is normal, otherwise, it is yellow.

P409_d5=0: the hydraulic chuck does not need the respond check.

P409_d3=0: the hydraulic chuck control signal is controlled by the level;

P409_d3=1: the hydraulic chuck control signal is pulse control; the pulse width is determined by the time of parameter **P327**.

P409_d1=0: the hydraulic chuck foot switch input is valid; **P409_d1**=1, the hydraulic chuck foot switch input is invalid.

◆ Execution process of chuck command:

In outer chuck mode, After M10 is executed, the system outputs the chuck clamping signal from M10 pin (the output pulse or the level signal is selected by the parameter) and the chuck clamping operation ends without needing the respond check signal; when needing the respond signal, the system waits the chuck clamping in-position; after it has checked the chuck clamping in-position signal (interface pin RM10 is at low level and RM11 is at high level) in the set time (P329: M responds check time specifying), otherwise the system prompts **"Alarm for chuck clamping respond check overtime";**

After M11 is executed, the system outputs the chuck releasing signal from M11 pin (the output pulse or the level signal is selected by the parameter) and the chuck clamping operation ends without

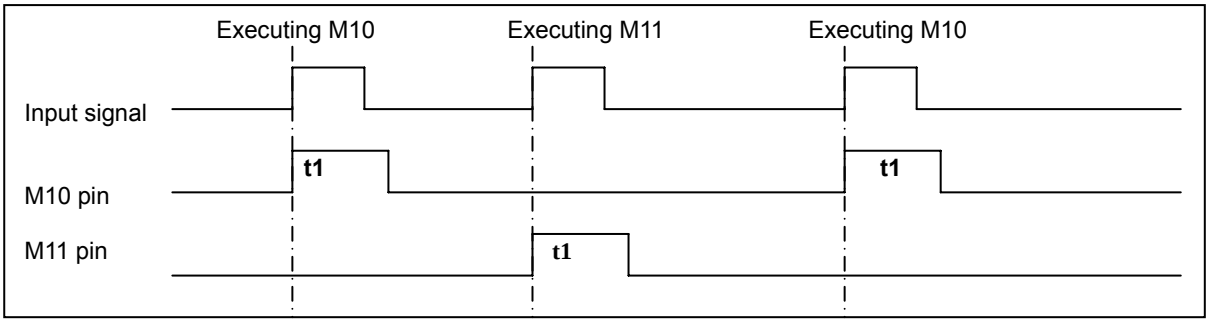
needing the respond check signal; when needing the respond signal, the system waits the chuck releasing in-position; after it has checked the chuck releasing in-position signal(interface pin RM11 is at low level and RM10 is at high level) in the set time (P329: M responds check time specifying), otherwise the system prompts **"Alarm for chuck releasing respond check overtime";**

In inner chuck mode: after M10 is executed, the system outputs the chuck clamping signal from M11 pin; after M11 is executed, the system outputs the chuck releasing signal from M10 pin, which is opposite to the output pin in the outer chuck mode, and others are the same.

Besides using commands, the external foot switch also can control the hydraulic chuck. The system switches the chuck clamping/releasing by M10/M11 when the foot switch is stepped once. "Chuck foot switch" releases before the system is switched from other working mode to JOG or AUTO working mode, otherwise, the system alarms.

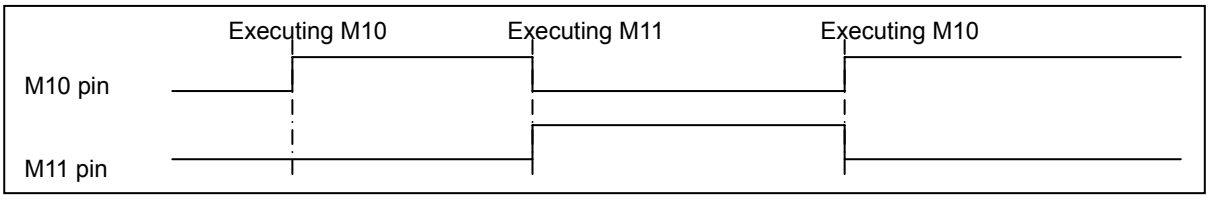
◆ Time sequence of hydraulic chuck control signal output:

1) M10, M11 output time sequence in pulse control mode:



t1：M10, M11 signal output hold time is set by P327 in pulse control mode;

2) M10, M11 output time sequence in level control mode:



【Note】

1) When the hydraulic chuck control is valid, the system defaults the chuck releasing after power on or emergency stop, the first control input of chuck is valid and the system outputs the signal of chuck clamping.

2) When it is the interlock protection between the chuck and the spindle: in the spindle running, the system forbids operating the chuck, otherwise, it alarms; in the chuck releasing, the system forbids starting the spindle, otherwise, it alarms.

3) In automatic continuous run, the foot switch operation is invalid no matter what the spindle rotates.

4) When the chuck operation is failure or interrupted (reset, emergency stop), the system takes the chuck is in the unconfirmed position, prompts the chuck flashing in red (M10 or M11), at the moment, the system cannot start the machining programs; the system recovers the

normal state when the chuck operation is executed once again or the system is turned on again.

5) The chuck respond signal consecutive check (P409d5=1)is to continuously check whether the chuck abnormally releases in the normal or machining state. If the above is set the alarm (**P402_d4**=1), the system stops the program machining and closes the spindle when the chuck releases in machining.

6) When the chuck signal cancels (M12), the chuck state (M10 or M11) is displayed with the underline, i.e.M10 or M11.

4.4.8 Hydraulic Tailstock Control Function

◆ Tailstock operation

In JOG working mode, input M78/M79 to control the tailstock forward/backward.

Input: M 7 8 ENTER ; the tailstock goes forward.

Input: M 7 9 ENTER ; the tailstock goes backward.

Input: M 8 0 ENTER ; cancel the tailstock control signal. (use M80 for the special tailstock device).

【Relative parameters】

P409_d4 is set to 0: the system has the hydraulic tailstock control function.

P402_d3=0: interlock between the hydraulic tailstock control and the spindle control.

P402_d2=0: the consecutive check of the hydraulic tailstock respond signal is close;

P402_d2=1: the consecutive check of the hydraulic tailstock respond signal is open

P409_d2=0: the hydraulic tailstock control signal is the level;

P409_d2=1: the hydraulic tailstock control signal is pulse control; the pulse width is determined by the time of parameter **P328**.

P409_d0=0: the hydraulic tailstock foot switch input is valid;

P409_d0=1: the hydraulic tailstock foot switch input is invalid;

◆Time sequence of execution process and signal output of tailstock command:

Define RM78 or RM79 in P519, P520 when the tailstock in-position signal is needed to check. After M78 is executed, the system outputs the tailstock forward signal from M78 pin (the output pulse or the level signal is selected by the parameter) and the tailstock forward operation ends without needing the respond check signal; when needing the respond signal, the system waits the tailstock forward in-position; after it has checked the tailstock forward in-position signal (interface pin RM78 is at low level and RM79 is at high level) in the set time (P329: M responds check time specifying), otherwise the system prompts **"Alarm for tailstock forward in-position respond check overtime";**

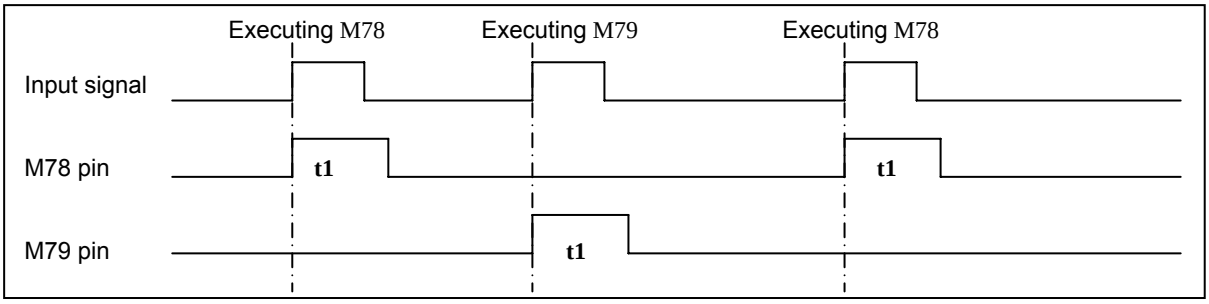
After M79 is executed, the system outputs the tailstock backward signal from M79 pin (the output pulse or the level signal is selected by the parameter) and the tailstock backward operation ends without needing the respond check signal; when needing the respond signal, the system waits the tailstock backward in-position; after it has checked the tailstock backward in-position signal (interface

pin RM78 is at low level and RM79 is at high level) in the set time (P329: M responds check time specifying), otherwise the system prompts **"Alarm for tailstock backward in-position respond check overtime";**

Besides using commands, the external foot switch also can control the hydraulic tailstock. The system switches the forward/backward by M78/M79 when the foot switch is stepped once.

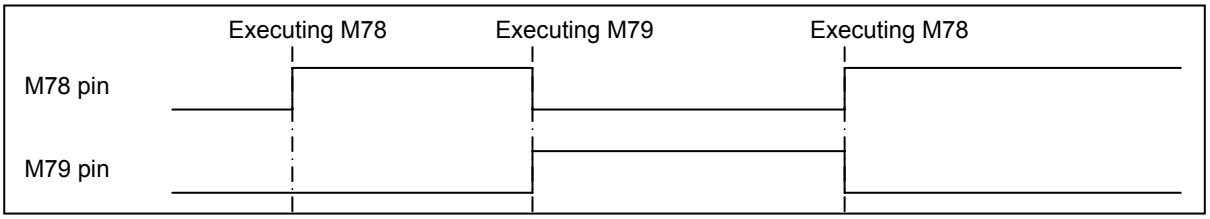
"Tailstock foot switch" releases before the system is switched from other working mode to JOG or AUTO working mode, otherwise, the system alarms normally.

1) M78, M79 output time sequence in pulse control mode:



t1：M78, M79 output hold time is set by P328 in pulse control mode;

2) M78, M79 output time sequence in level control mode:



【Note】

1) When the hydraulic tailstock control is valid, the system defaults the tailstock backward after power on or emergency stop, the first control input of chuck is valid and the system outputs the signal of tailstock forward.

2) When it is the interlock protection between the tailstock and the spindle: in the spindle running, the system forbids operating the tailstock, otherwise, it alarms.

3) In automatic continuous run, the tailstock control input is invalid no matter what the spindle rotates.

4) When the tailstock operation is failure or interrupted(reset, emergency stop), the system takes the tailstock is in the unconfirmed position, prompts the tailstock flashing in red (M78 or M79), at the moment, the system cannot start the machining programs; the system recovers the normal state when the tailstock operation is executed once again or the system is turned on again.

5) The tailstock respond signal consecutive check is to continuously check whether the tailstock abnormally releases in the normal or machining state. If the above is set the alarm (**P402_d2=**1), the system stops the program machining and closes the spindle when the chuck releases in machining.

6) When the tailstock signal cancels (M80), the tailstock state (M78 or M79) is displayed with the underline, i.e. M78 or M79.

4.4.9 Other Option Functions

Option function is the non-standard pin input/output control.

When the system needs some function, the operator defines its pin in the interface parameter and correctly connects with the wirings. The detailed interface parameter definitions are referred to ***OPERATION 4.6 Parameter Working mode***; the detailed wiring connection is referred to ***CONNECTION Chapter 3 CNC Device Connection***.

【Warning】

Pin definition must be performed by the machine manufacturer; the improper definition maybe damages the system and the machine electricity.

4.4.9.1 Three-Color Indicator Control

When the system needs the function, the operator should define its output pin in the interface parameter and correctly connects with wiring; the system output the signal in the corresponding pin.

P502：LMP3：Green lamp (program run signal indicator 3);

P503：LMP2：Yellow lamp (Alarm lamp control signal 2);

P504：LMP1：Red lamp, alarm lamp (Alarm lamp controls signal 1).

[Functional description]

1) The tricolor light control is valid in JOG /AUTO working mode.

2) The green indicator light means the program normally runs.

3) The green indicator goes out and the red one lights when the system alarms.

4) The red and green indicators go out and the yellow indicator lights when the program stops running without alarm.

4.4.9.2 Lubricating Control

When the system needs the function, the operator should define its output pin in the interface parameter and correctly connects with wiring; the system output the signal in the corresponding pin.

P506：M32O： Lubricating controls the output signal.

[Functional description]

1) Non-auto lubricating:

P330 is set to 0: non-automatic lubricating is controlled by the command about the lubricating ON/OFF.

In JOG /AUTO working mode, input lubricating ON/OFF M32/M33 is valid.

After M32 is executed, lubricating outputs; after M33 is executed, the lubricating output is cancelled.

2) Automatic lubricating:

It is the automatic lubricating when P330 is set to other than 0.

The system can set Lubricating starting time and Lubricating interval time. After the system is turned on, it executes the lubricating for the time set by P330, then stops the output, executes the cycle lubricating after the set time set by P331.

[Note]

When the system starts the automatic lubricating function, P330 and P331 values are more than 1s; when they are less than 1s, the system takes them as 1s.

4.4.9.3 Machine Electricity Delay Power-on Control

When the system needs the function, the operator should define its output pin in the interface parameter and correctly connects with wiring; the system output the signal in the corresponding pin.

P505：MDLY： Machine electricity delay power-on control signal.

[Functional description]

When machine electrical power-on delay control signal is defined in interface parameters, the system will wait 3s delay time after power-on, and outputs that signal from self-defined pins. The operation keys are invalid during the 3 seconds.

4.4.9.4 External MPG Operation

When the system needs the function, the operator should define its output pin in the interface parameter and correctly connects with wiring; the system output the signal in the corresponding pin.

Relative parameter: when **P400_d1** is set to 1, the external MPG control knob is valid, Z/X/Y axis option key, step width regulation key are invalid.

In JOG working mode, press MPG to switch to MPG mode, the indicator lights and the system displays the external MPG control knob, axis option knob and movement knob state. The coordinates of selected coordinate axis is displayed in highlight.

Simultaneously display the external MPG icon on the screen.

1) Axis option knob of MPG:

WsZ external MPG axis option Z: select Z when it is connected;

WsX external MPG axis option X: select X when it is connected;

WsY external MPG axis option Y: select Y when it is connected;

When WsZ , WsX , WsY are not connected, the system cancels the coordinate axis in the highlight state; the MPG is invalid.

2) Each movement knob of MPG graduation:

Wbk1 external MPG override1: when it is connected, the system selects 0.100 gear; (when the parameter setting the gear is invalid, it is 0.001 gear);

Wbk2 external MPG override 2: when it is connected, the system selects 0.010 gear;

Fail to connect the Wbk2 and Wbk1, the system selects 0.001 gear;

3) MGP emergency stop button:

Wsp external MPG emergency stop signal: the system should use the normally-closed contact; the function is equal to the system ESP signal.

4.4.9.5 Safety Door Check Function

When the system needs the function, the operator should define its output pin in the interface parameter and correctly connects with wiring; the system output the signal in the corresponding pin.

P511：SAGT： safety door check signal.

[Functional description]

1) SAGT is connected with 0V, CNC confirms that the safety door closes;
2) In AUTO Working mode, the system alarms "Alarm for safety door not be closed" when it has checked that the safety door opens.
3) In automatic run, when the system has checked the safety door has opened, the axis feed stops, the cooling closes and the system alarms;
4) The safety door check function is valid in AUTO working mode.

4.4.9.6 Pressure Low Alarm Check Function

[Relative parameters]

P412_d5=1: pressure low check function. **P412_d4** sets pressure low alarm level; **P412_d4**=1: low level alarm, **P412_d4**=0: high level alarm.

P332 sets the durable pressure low alarm time.

[Functional description]

When the system selects the pressure low alarm check function, it displays the press check icon at right-hand side in JOG Working mode and AUTO working mode, the icon is green solid triangular▲ when the pressure is normal. Once the system has checked that pressure low alarm signal PRES is valid, the durable pressure low alarm time is not more than the half of the set time by P332, the icon is a yellow hollow triangular△; when the pressure low durable time exceeds the half, the icon is a red hollow triangular △ and the system alarms "Pressure low alarm"; at the moment, the axis feed pauses, the spindle stops rotating and the automatic cycle cannot start;

4.4.10 Searching Run Message in JOG Working Mode

The detailed is referred to **OPERATION**, AUTO working mode 4.5.8.

4.4.11 Appendix:

4.4.11.1 MDI Input Controlling M Command Table MDI

◆ Executable M commands in JOG working mode:

Sort	Command	Function	Remark
Spindle control	M03, M04, M05	CW, CCW, stop	Function interlock, state keeping
Cooling ON	M08, M09	Cooling ON, OFF	
Chuck	M10, M11, M12	Clamping, releasing, canceling chuck output signal	
Spindle	M13, M14	Output/Retraction spindle clamping signal	
Lubricating	M32, M33	Lubricating ON, OFF	

Tailstock	M78, M79, M80	Tailstock forward, backward, cancel tailstock output signal	Function interlock, state keeping
Operator output 1	M21, M22		Function interlock, state keeping
Operator output 2	M23, M24		Function interlock, state keeping
Spindle gear	M41, M42, M43, M44	Spindle gear shifting gear 1, 2, 3, 4	Function interlock, state keeping
Operator customized command	M50 ~ M74		
Output control and check	M82	E.g. M82 Q17.0 D3 or M82 Q17.0	
Set the spindle working mode	M47, M48		
Spindle orientation control	M87、M88		

Note: When the operator inputs M command and the first digit is 0, it can be omitted. The command functions are the same those of AUTO Working mode. The detailed is referred to PROGRAMMING.

4.4.12 Spindle Turn Function

Manual tapping function is to manually turn the spindle and the selected coordinate axis links along the spindle to realize the tapping and thread run-out when the spindle stops.

In JOG working mode, the spindle stops stably. Press * and the system enters the spindle turn function state and prompts: “Inputting tapping axis (X/Y/Z).”

After the movement axis is selected by X/Y/Z key, the system prompts “Input the tapping pitch (mm):”, and then enter the manual tapping state after inputting the pitch by “ENTER”. In this case, the spindle can be rotated manually, and the tapping axis can be performed axis movement depending on the spindle rotation.

In manual tapping state, press ESC to exit the manual tapping state, the motion axis decelerates to stop when the tapping axis exits from the motion state.

[Functional description]

The function is valid in JOG working mode, the tapping in manual tapping state moves along the spindle rotation.

The axis motion speed is determined by the spindle speed and the pitch, the axis motion direction is determined by the pitch sign as follows:

When P is positive and the spindle turns CCW, the coordinate axis moves negatively; when the spindle turns CW, the coordinate axis moves positively

When P is negative and the spindle turns CCW, the coordinate axis moves positively; when the spindle turns CW, the coordinate axis moves negatively.

[Note]

1）When the speed in tapping is too fast or there is the limit alarm, the system automatically exits the tapping and alarms.

2）The pitch P is expressed with the metric, range: 0.001mm～500.000mm(the negative sign is added to the front of the range, i.e. “right-hand” or “left-hand” tapping) .

3）Z/X/Y axis manual tapping speed is restricted by parameter P100, P101, P102 respectively.

4.5 AUTO Working Mode

In **AUTO** working mode, the relative setting or input format and example descriptions are as follows: the required input letter key, or digit key are expressed with the underline; the system prompts the message is expressed with the frame.

In AUTO working mode, the system displays at the top right. Press it and the system pops-up the operation key catalog in AUTO working mode; press it and the window closes; directly press other function keys, and the window automatically closes.

Press

and the system enter AUTO working mode. The system completes the part machining of the specified machining program in AUTO Working mode; the system runs from the first line of the selected workpiece program, and gradually executes till the program ends.

The system combines the operator parameter table, offset value to analyze and pre-inspection the part programs. When the system detects the problem in advance, executing the machining program causes the **serious result** and the system closes the CYCLE START key. In the condition, pressing CYCLE START key is invalid and the system refuses to execute the program; the system can execute after the program or the parameter is modified according to the alarm message.

The system provides many part program execution modes, and the operator must set before running to get the safety of machining process.

◆ Main functions in AUTO Working mode:

1) Set **SINGLE/CONTINUOUS** run program
2) Set **DRY RUN (without output)** check run, and the system **accelerates to execute** the program in DRY RUN mode.
3) Check the **software limit alarm in advance** before running programs
4) Set **blocks** and execute the middle of the program
5) Spindle, cooling press key control
6) Execute machining programs by **pause, block stop, end stop**, **cycle stop**
7) Tune **cutting speed override** proportion
8) Correct offset in execution process
9) Real-time state display of machine, pop-up window real-time alarm

◆Display content on screen as Fig. 4-5:

Upper Top: display the execution mode (**SINGLE/ CONTINUOUS**, **DRY RUN**), current program number, workpiece count, machining time; system function operation method prompt key ;

Left top: display tool nose coordinates and machine coordinates, or tool nose path graph;

Left bottom: display machining block (pointer points to the current block);

Right middle: display the current state of machine, including spindle, cooling, lubricating, tool post, chuck, tailstock, speed, cutting speed and so on;

Pop-up window: display alarm message of execution program.

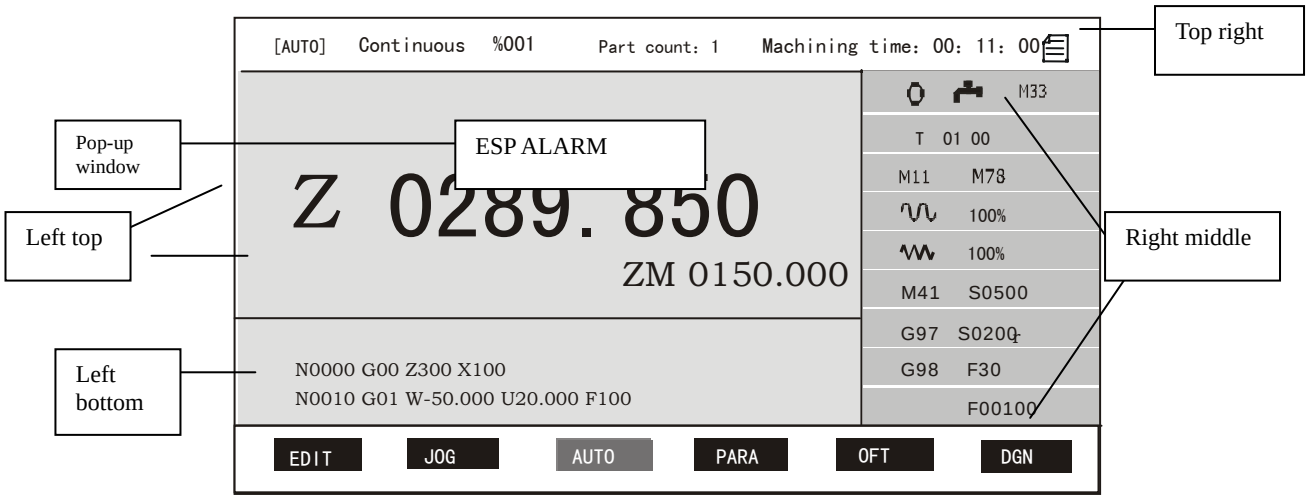


Fig 4-5 AUTO working mode

4.5.1 System Working Mode in AUTO Working Mode

In AUTO working mode, the system is in several mode according to execute workpiece programs; when the system is in different states, it permits the functions are different; there are several states as follows:

Initial state: it is the execution pointer of the program points to the first line but the system has not executed; the system is just now switched from other mode to AUTO working mode to enter the initial state or returns the initial state after the program is executed or the system alarms.

Run state: the system is executing the block and the coordinate axis is moving.

Pause state: the current block has not executed completely in the course of executing the axis motion command to pause; the system waits the operator presses the key to execute the operation.

Block stop state: the current block has been executed and the next has not executed, the system waits the operator presses the key to execute the operation.

4.5.2 Function Key Operation in AUTO Working Mode

4.5.2.1 SINGLE Execution and CONTINUOUS Execution Switch

SINGLE execution and CONTINUOUS execution switch:

Press

and the system switches **SINGLE/CONTINUOUS** circularly; (It is enabled in any states).

In continuous execution, press the key and the system switches to SINGLE working mode, after the current block is executed, the system stops, and continuously executes after CYCLE START is

pressed.

In CONTINUOUS working mode, press CYCLE START, and the program is executed from the beginning to the end.

In SINGLE working mode, press CYCLE START key once and the system executes one block (for the cycle command, the system only executes one operation; press CYCLE START, and the system executes one operation).

4.5.2.2 Dry Run and Machining Run Switch

Check all content of machining programs in dry run, which can ensure the machining workpiece is not discarded because of some programming data error in the program.

Dry run/ machining run switch:

Press

, and the system switches **dry run/automatic machining run** mode

In DRY working mode, whether M, S, T is valid is set by the parameter, coordinates of each axis automatically recover the previous before the Dry working mode.

Relative parameters in DRY Run working mode:

P401_d7：

0：When the system executes the miscellaneous command, it must output the signal, check the signal, which is the same that of the normal machining.

1：The system does not output the signal or check the signal when the system executes the miscellaneous command.

P401_d6：

0：The execution speed of feed command is set by the program, which is the same that of the normal machining.

1：The execution speed of feed command is not set by the program, max. speed (P113) of cutting feed displays the program path.

[Note]

1) The dry run key is valid when the program is executed in the initial state. In the course of program execution, the key is invalid and cannot be switched when the program does not end and the system has not exited the execution state.

2) **P401_d7=0**: in DRY RUN working mode, all miscellaneous command M, S, T are executed; the system recovers to the previous state when it exits from the dry run state.

3) **P401_d7=1**: in DRY RUN working mode, the system does not output and check the signal when it executes the miscellaneous function; when the system executes T function, the tool offset number is executed (when the previous is T11, it becomes T13 after T33 is executed), the system recovers to the previous after it exits from the DRY RUN working mode.

4) In DRY RUN working state, all macro command and M50～M74 are normally executed; after the system modifies the offset and the system exits from the DRY RUN working mode, the tool nose coordinates of corresponding tool offset number are changed.

5) The workpiece counter does not automatically add 1 in DRY RUN working mode.

4.5.2.3 Running a part Program from the First Block

After entering AUTO working mode, the system enters the initial state, and the program pointer points to the first block of the current program, and CYCLE START key is pressed to start the program to automatically run.

The being executed block displays and flashes in poor color; the first line is the executed block, and the 3rd line is to be executed; when the machining program is the conditional command, the skip or call target is not well-defined, and the 3rd line may not be displayed.

4.5.2.4 Running a Part Program From a Specified Block

In some special conditions, it is necessary to start to run from some block in a part program. This system allows starting any one block of current part program. (It is valid in initial state)

Steps to select a block:

1) Press INPUT, and the system pops-up the program browse window, displays the current program and the pointer points to the first block of program.

2) Pressing , and the system displays the content of the top(down) block or up(down) page. Press ESC and the system exits from the selected and displays the previous block.

3) When the pointer points to the required block, ENTER is pressed and the system prompts "Run?" to wait the next execution.

4) At the moment, press CYCLE START and the system executes the program from the block pointed by the pointer; press ESC and the system exits the selection and the pointer points to the first block.

【Note】

1) The specified block cannot be in canned cycles, compound cycle bodies or subprograms, otherwise there is the unexpected run. The system selects G00 or the tool change command before G00.

2) When the system runs the program from the specified block, the selected block should be the linear movement or S. M. T.

3) In the course of program execution, press INPUT and the system pops-up the browse and forbids executing the selected block.

4.5.3 Displaying in a Part Program Running

When the part program is running, this system displays the running state, the dynamic run coordinate, the workpiece planar solid graph, and the path of tool nose in the course of program running, which is very convenient to monitor the running state of the machine and the program. The

display as follows:

- The dynamic coordinates or the dynamic tool nose movement path graph or workpiece contour graph.
- Current block content.
- Spindle, cooling, lubricating, tool, speed, chuck, tailstock, and machine miscellaneous function state.
- Feedrate override, rapid override.
- Machining time, workpiece count.

4.5.3.1 Machining Workpiece Count and Timing

Workpiece count: When the program being executed once means the program ends (M02, M20, and M30), the machining quantity count adds 1, and max. count range is 99999, and the count becomes 0 when it exceeds the max. When the program ends with M99, the machining quantity also increases by 1 after the program is executed once.

Machining time: Record machining program execution time. When CYCLE START is pressed and the system executes the program, the timing does not end till the program ends. In running, the system pauses; at the same time, the timing stops. In SINGLE working mode, the system only records the run time of each block. The system displays max. machining time range: 99 hours, 59 minutes and 59 seconds. When the machining time reaches the max. value, It automatically becomes zero and continuously runs program and executes the timing again.

Workpiece count and machining time clearing: in initial state, continuously press

twice and the workpiece count clears; continuously press twice, and the machining time clears.

4.5.4 Manual Operation of Miscellaneous Function

In AUTO working mode, press function keys to execute some miscellaneous function operation of the machine, and the other functions are the same in JOG working mode.

1) Cooling ON/OFF is valid in any states.

2) When **P400_d5** is set to 0, spindle CW, stop key are valid:(invalid in run state).

3) In initial state, when the hydraulic chuck control function is valid, the external button or foot switch controls the hydraulic chuck clamping/releasing which interlocks with the spindle.

4) In initial state, when the hydraulic tailstock control function is valid, the external button or foot switch controls the hydraulic tailstock forward/backward which interlocks with the

spindle.

4.5.5 Speed Override Tune in AUTO Working Mode

4.5.5.1 Speed Override Tune

In AUTO working mode, the feedrate override and rapid override keys in any state are valid.

In AUTO working mode, the system can set the speed without changing the program and parameter. Change the program running speed by changing the speed override.

● **Feedrate override** Speed word F setting value in tuning the program:

Actual feedrate speed = Fx feedrate

The feedrate override has 16 gears 0%～150% (increment of 10%), all commands controlled by the feedrate is controlled by the feedrate override.

● **Rapid override** G00 rapid traverse command speed in tuning programs.

Z (X or Y) actual rapid traverse speed = P100 (P101, P102) ×rapid override

The rapid traverse override is divided into **25%, 50%, 75%, 100%**. All rapid traverse commands and the operations are controlled by the rapid traverse override.

Whether programs are running or not, pressing rapid override +/-, feedrate override +/- keys can change the speed override. The actual traverse speed of the machine slider changes if the speed override is changed when the programs are running.

In program running, the program stops running when the feedrate override is 0, the system prompts: **Feedrate override be zero.** The program continuously runs when the feedrate override is not zero.

4.5.5.2 MPG Speed Control

In AUTO working mode, using MPG controls rapid/feedrate override. **P402_d1**=1: MPG controlling rapid/feedrate override is valid, and the actual feedrate and the rapid speed are as follows:

Actual feedrate= F x feedrate override x MPG override

Z (X, Y) actual rapid speed = P100 (P101, P102) x rapid override x MPG override

MPG override range：0% ~ 100%.

MPG controlling rapid/feedrate override:

◆Method 1:

When parameter **P402_d0=0**, the system enables MPG control rapid traverse/feedrate override in method 1; the system set the current position (A point) to 0% of MPG override. From point A to 100 scale, every 1 scale in CW direction means an increase of 1% of MPG override; every 1 scale in CCW direction means an decrease of 1% of MPG override.

◆ Method 2:

When **P402_d0**=1, the system enters MPG controlling rapid/feedrate override according to method 2; when MPG override is set 0%, MPG override changes along MPG CW rotation, MPG CW rotation becomes gradually fast, MPG override is regulated 0%~ 100%. To avoid machine to be impacted by uneven speed of MPG, and the regulation range is less than 10% every time. MPG stops

after it CW rotates, MPG override becomes 0%. After MPG CW rotates and rotates CCW (or CCW rotates and then stops), MPG override keeps the CCW instant override till MPG CW rotates (CW rotates and then stops).

MPG controlling rapid/feedrate override explanation:

In AUTO working mode, before the machine program is not executed or the program pauses, single block stops, the cycle stops or the feed holds (including spindle/feed hold), 【**MPG**】indicator lights after pressing it means the system is in MPG controlling rapid/feedrate override. At right top, the yellow means MPG control mode and current MPG override. The system automatically cancels MPG controlling rapid/feedrate override mode after each program is executed.

【Note】

In the thread machining commands (G32, G33, G34), the feedrate is determined by the spindle speed instead of F value, and MPG override is invalid here.

4.5.6 Interference Operation in Program Execution Process

4.5.6.1 Press Key Interference in Program Execution

Interference operations in program execution:

ESP: Immediately stops not continuously start the execution.

DWELL: Press CYCLE START to continuously execute the program.

SINGLE BLOCK STOP: Press CYCLE START to continuously execute after the block is completed and the system stops.

CYCLE STOP: Press CYCLE START to continuously execute the program after the cycle is completed and the system pauses.

◆ Pause

1) Press CYCLE PAUSE and the system pauses in executing the command. After the system responds, each motion axis decelerates to stop, which **Pause** is displayed on the bottom-left.
2) In pause state, press CYCLE START, and the system recovers the program to continuously execute the left; press ESC, the program exits and the system returns to the auto Initial state, and the pointer points to the first block of the current program.

【Special notice】

1) After the pause, the system can control the spindle, the chuck and the tailstock; before CYCLE START is pressed, ensure the spindle is started, the chuck and the tailstock have been ready, otherwise, which maybe damage the machine and hurt the persons.
2) When the system follows the blocks for the spindle machining thread in executing G32, G33, G34, the press key is invalid.
3) For details about the pause or single block stop function, refer to Section 8.2.2 “Explanation of Customized Command Storage” in OPERATION.

◆ Single block stop

1) In continuously executing the program, press SINGLE and the system is switched to the single block execution mode, and when the current block is executed, the system displays **Single block stop**.

2) After the single block stops, press CYCLE START and the program continuously runs. Press ESC and the system returns to the auto initial state and the pointer points to the first block of the current program.

【Note】

In executing the fixed cycle command, the single block stop is valid after each step of the fixed cycle is completed.

◆ Cycle end stop

In continuously executing the program, press **X/Y/Z axis shift key** and then the system displays **CYCLE STOP: ON**; and the system displays **CYCLE STOP** after M20 is executed.

4.5.6.2 External Feed/ Spindle Hold Knob

The external feed**/** spindle hold knob is valid in AUTO working mode.

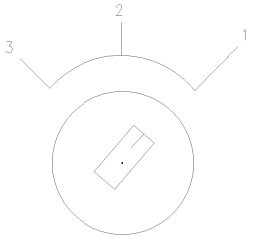
Whether the external feed**/**spindle hold knob is valid is controlled by **P412_d6**.

P412_d6=1: the system external feed**/**spindle hold knob is valid; the input signal is received by MXZ1, MXZ2.

P412_d6=0: the system external feed**/**spindle hold knob is invalid; the pin of input signal can be used as others.

◆External feed/spindle hold knob introduction:

The system has an external interface of feed/spindle hold knob. Move or stop the spindle and the slider when the knob is placed on the different position. Use the knob to control conveniently the starting/stopping of spindle and the slide in debugging the program. There are three positions of feed hold knob and its function as follows:



Note: see the specific symbol specification of the external feed/spindle hold knob

Feed hold knob

Position 1: permit the spindle to rotate and the slider to traverse.

Position 2: permit the spindle to rotate and forbid the slider to traverse.

Position 3: forbid the spindle to rotate and the slider to traverse.

◆External feed/spindle hold knob use:

Before program running:

When the external feed/spindle holding button is on the position 1, 2 and 3, the spindle start/stop can be controlled manually.

In SINGLE working mode:

When the External feed/Spindle hold knob is located at position 1, all the commands are executed as usual; when it is at position 2, the spindle control commands can be executed rather than the move commands of X, Z axes. These move commands can be executed only when the knob is rotated back to position 1. When it is at position 3, no block can be executed. To execute the blocks, rotate the knob back to position 2 or position 1.

In CONTINUOUS run mode:

After the system starts programs, the feed hold knob can be rotated any time to control the spindle and the slider.

When the knob is placed to the position 1, programs run normally.

When the knob is rotated from 1 to 2, the slider stops and the spindle still keeps the previous state.

When the knob is rotated from 2 to 3, the spindle stops.

When the knob is rotated from 3 to 2, the spindle recovers the previous state.

When the knob is rotated from 2 to 1, the slider starts to run.

The system will automatically exits from the auto initial state after ESC or RESET is pressed in the course of the feed hold and the spindle stopping. The previous state of spindle and the unfinished commands cannot be reserved. Programs are restarted if the machining is executed continuously.

4.5.6.3 External Start and Pause Signal

The “External cycle start” described in this manual is called the “External cycle start-up”

The external cycle start/dwell signal is controlled by parameter **P412_d7.**

P412_d7=1:The system external start/dwell signal is enabled; the input signal is led into by **ST, SP** pin.

P412_d7=0: The system external start/dwell signal is disabled; the pin of input signal can be used as others.

The external pause operation key signal (SP) has the same function with that of the feed hold key (cycle pause key) on the system panel; the external cycle start key signal (ST) has the same function with that of the cycle start key on the system panel. SP, ST are input to the system from the machine, and they are valid when the low level is connected.

Before the system is switched from other working mode to AUTO working mode, “External start button” and “External pause” button are released (power-off), otherwise, the system alarms.

The detailed circuit connection method is referred to **CONNECTION, Chapter 3 CNC Device Connection**.

4.5.6.4 Feed Device Alarm Function

When the system needs the function, the input pin is defined in the interface parameter and is correctly connected; the system checks the signal on the corresponding pin.

P512： Dalm： Feed device alarm check signal.

【Functional description】

1) When the system checks the signal in M20, it automatically stops and alarms.

2) If the EOB is M02 and M30 commands instead of M20, and the then feed equipment alarm function is disabled.

4.5.7 Modifying Offset in Program Run

The system pops-up the window to modify the offset value in machining and the operator must be careful.

4.5.7.1 Modifying Offset Method in Program Run

◆ Modifying offset method:

1) In automatic running, press OFFSET and the system pop-up the window to display the modifying offset, press it again and the system closes the window.

2) Press , or , to select the required modification tool offset number; press and to select the required modification tool offset, and the detailed operations are referred to **OPERATION, 4.7 OFFSET Working Mode.**

3) Press INPUT to input data. Press ENTER to replaces the previous data. Press ALTER to add on the previous data. Press ESC to cancel the input data.

4) Press OFFSET or press ESC to exit the offset display window after the input is completed.

【Note】

1) During data inputting, you can switch to AUTO mode display page, and when you switch back, the original input data that have been stored by pressing key ENTER remain the same, you can continue to input data.

2) In offset display window, pause, modifying feedrate override operations are valid in AUTO working mode. In pause, the system can be switched to the offset display window to modify the offset.

4.5.7.2 Modifying Tool Compensation Validity in Program Running

[Note]

The modified offset data is valid when the system executes the tool change. When the modified is the offset data corresponding to the current tool offset number, the modified value is valid in the next tool change. When the modified is the tool compensation value corresponding to the unexecuted tool offset number, the modified value is value in this execution.

When the program has no the tool exchange command but the system has modified the offset value current tool offset number, after the system executes M02, M30, M20, “RESET” and stops, the modified offset is valid.

4.5.8 Searching Run Message in AUTO Working Mode

The function is valid in any states in AUTO and JOG working mode.

In automatically machining part programs, the system pops-up the window to search the macro variable, I/O variable and others in running process as follows:

Macro variable: search all common variable used in the program, and modify the common variable value;

I/O variable: search the interface variable value (i.e. the system interface state);

Others: search the executed block quantity, the spindle wave range in the thread machining, program nested call layers and program cycle in executing subprogram.

◆ operation method as follows:

1) In automatic running, press and the system pops-up the window to display the variable, I/O variable and others), press ESC again and the system closes the window.

2) Press , to select the required searching items and the selected item is displayed in black.

3) When there are many variables are searched, press to select the variable(it is pointed by the pointer), at the moment, the macro variable cannot be changed; press or to search the front or the latter one macro variable; press or to search the variable in the page up or page down with 7 lines in each page; press to exit, at the moment, the macro variable is displayed in black.

4) In Auto initial state, the common variable r100~r199 pointed with cursor can be alternated by the ENTER key.

5) Press ESC to exit the display window.

【Explanation 】

1) Macro variable: display the running common variable edited to the program, including variable number, variable name, variable value and state. The variable value is displayed in the dynamic along the program changing in running, the number and variable name are sorted from the small to the big. Variable r001～r040 are displayed in brown, r041～r99 in orange, r100～r199 in green.

2) I/O variable: display in dynamic the interface state of current running program. Two kinds of state value of input interface variable: 0 (LOW level) or 1 (HIGH level); the external signal valid and the pin is connected with 0V in LOW; it is invalid in HIGH. The out interface variable state: when the system output “0”, the external forms the conductive loop; when the system output “1”, the system is in high-resistance, and the external cannot form the conductive loop.

r1001～r1032 are the input interface states; r2001～r2032 are the output interface states; 8 groups to display, the first line displays the first group: r1008, r1007, r1006, r1005, r1004, r1003, r1002, r1001 and others is in order reason by analogy.

3) Others: search the executed block quantity, the spindle wave range in the thread machining, program nested call layers and program cycle in executing subprogram.

4) Before the system does not start the machining program, macro variabler100～r199 can be modified manually, the modification method is to select the required modification macro variable, the operator presses ENTER to input the value to change, presses ENTER again and the modification is completed.

4.5.9 Program Reference Point Return in AUTO Working Mode

The function is valid in initial state of AUTO working mode

After the system sets the workpiece coordinate system and program reference point, it starts the machining program when the machine slider stops any position. At the moment, the first movement command of the machining program must be in G00, must execute X, Z absolute coordinate positioning. In the condition, press program zero return or G command and the system can return the set program reference point. After G command is executed, the machining is continuously executed, the system must use G00 two-axis absolute coordinate to position simultaneously to get the correct machining.

After the system returns to the program zero by manual press key, it automatically points to the first block of the program. At the moment, press CYCLE START and the system starts running the fist block.

4.5.10 System Reset and ESP Signal Processing in AUTO Working Mode

In Auto operation mode, the system enters the Reset state by pressing Reset key. See Section 4.1.4.7 “Reset Operation” in Part Ⅰ Operation.

For details about the emergency stop signal in Auto operation, see Section 4.1.4.3 “Emergency Stop Alarm” in Part Ⅰ Operation.

[Note]

1) Before releasing the emergency stop alarm, first confirm the fault has been removed;

2) Pressing down the emergency stop button before Power On and Power Off can reduce the surging to the equipment.

3) Re-perform machine zero return after releasing the emergency stop alarm, to ensure the coordinate position is correct (if the machine is not equipped with the machine zero, it is forbidden to perform machine zero return);

4) The external emergency stop is valid only when bit parameter P404_d7 is set to 0.

4.5.11 Regulating LCD Brightness in AUTO, JOG Working Mode

The function is valid in the initial state in AUTO working mode.

◆ Operation method as follow:

1) Press continuously 9 twice and the system pops-up the brightness regulation window, any key except for the brightness regulation is pressed and the system closes the window.

2) The brightness regulation window has 0～10 grade: 0 grade is the darkest, 10 grade is the brightest; press the brightness regulation key increases LCD brightness, press to reduce LCD brightness.

[Note]

1) When the LCD has LED backlight, the brightness adjusting function is valid for the LCD screen; when the LCD has CCFL backlight, the function is invalid for the screen.

2) There is no operation in it when the brightness regulation window closes automatically in 10 seconds.

3) LCD brightness can be regulated in JOG working mode, and its regulation method is the same as that in AUTO working mode.

4.5.12 Display of M Command Execution State in AUTO, MANUAL Mode

The displayed M commands are divided into following 11 groups: M21/22, M23/24, M51/52, M53/54、M55/56, M57/58, M61/62, M63/64, M65/66, M67/68 and other M commands: M50, M59, M60, M69~M74, M81/M82/M83/84, M13/14. In MANUAL/AUTO mode, when the above mentioned M commands are executed, corresponding prompts will be displayed on the screen. Red means M command is being executed; green means finished; yellow means suspended.

4.5.13 Operations in AUTO Mode

In AUTO mode, when spindle and chuck are interlocked, if M3 is executed to turn ON the spindle, there are two choices after alarm occurs:

1) Press ESC to exit to the initial state of AUTO mode;

2) Use foot switch to execute M10; press CYCLE START to turn on the spindle (M3); press ESC to exit to the initial state of AUTO mode.

4.5.14 The External Feedrate Knob in Auto, Manual Mode

The knob of external feedrate is enabled in the Manual/Auto mode. If this function is desired, and its inputted pin should be defined in the interface parameter; at the same time, correctly connect the

wiring. The system will inspect this signal based upon the corresponding pin.

Whether the external feedrate knob is enabled is controlled by parameter **P400_d0.**

P400_d0=1 means that the system external feedrate knob is enabled; the input signal is lead-in by Wfk1~Wfk4, Wf_H, Wf_P pins.

P400_d0=0means that the external feedrate knob is disabled; the input signal pin is only for other purposes.

The external feedrate knob is same to the feedrate button function in the panel, the 0~150% 16-level real-time adjustment can be performed. During the program procedure, it is very convenient to control the overall feedrate commands in the program by use the knob.

Refer to the *CNC Equipment Connection* of Chapter Three in CONNECTION for the detailed circuit connection method.

Related parameter:**P400_d0、P561~P566.**

4.6 Parameter Working Mode

PARAMETER working mode function includes parameter input, parameter draw, parameter solidifying; the system prompts intellectively each operation. At the same time, the operator can press hp1 at the top right to learn the parameter operation key catalog.

The relative settings, operation input formats and example descriptions are as follows: all required function keys are expressed with icons; all input letter keys, or digit keys are expressed with underline; the system prompting message is expressed with frame.

Press to cancel the mistaken input when the input letter or digit is wrong.

Press

to exit the current operation before confirmation when the operator executes some setting or some operation or man-machine dialog.

Note: before the operator modifies the parameter, all parameter setting values in the system must be saved (save them to personal PC). Once the parameter is changed by mistake or the system, the system can recover by the saved data.

The system classifies the parameters: reference point coordinate parameter, motion parameter, transmission parameter, miscellaneous function parameter, interface parameter, variable initial value parameter and pitch parameter.

Press

to enter PARAMETER working mode. (The system pops-up the window to require inputting the password; the operator inputs the password or directly presses ENTER to enter the parameter window).

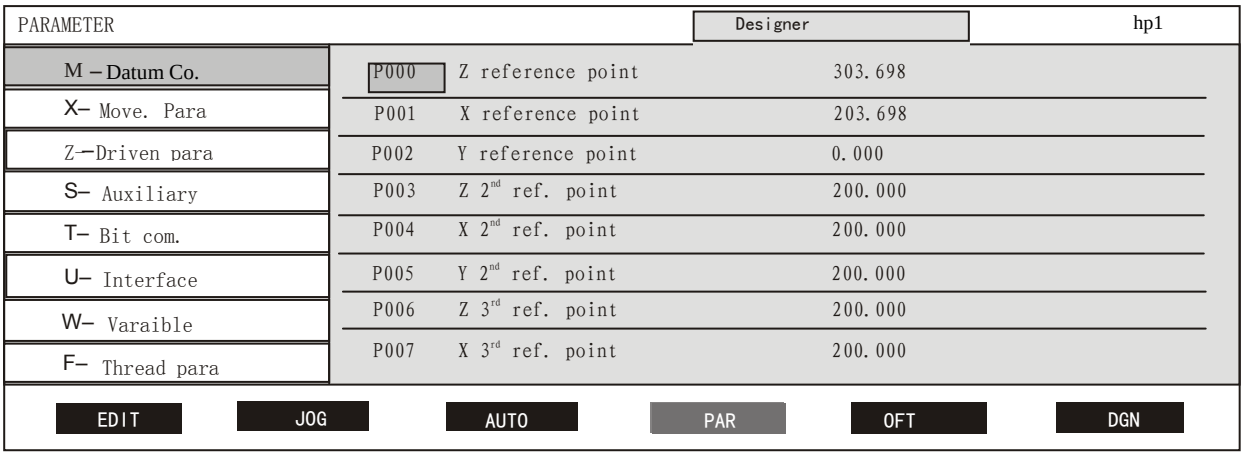


Fig. 4-6 parameter working mode

4.6.1 Parameter Overview

Parameter operation characteristics including:

1) Press the parameter password level input by the operator to modify the corresponding level parameter.

2) When the operator directly presses ENTER instead inputting the password, the operation level is 4 to enter the parameter window in which the operator only reads but cannot modify the parameter.

3) For the parameter input and display format, the decimal must has the decimal point, and the negative number must has the negative sign; the system limits the valid digits to get the convenient operation and using safety.

4) The operator can open the prompt message window of parameter data input range.

5) The system automatically checks the parameter data after power-on and automatically prompts the initialization when it finds out the data in disorder.

6) The system sets the applicable safety parameters and the operator can reduce the accidences caused by the mistaken operation by the proper parameter setting.

4.6.1.1 Parameter Privilege

The parameter privilege is to modify the parameter password level. To get the convenient management, the system provides the parameter privilege setting function, the current operation level is displayed on the top prompt bar in the parameter window.

Parameter password level settings from high to low are as follows:

1 level: *** set by the machine manufacturer, can modify the parameter range: parameter level >=1;

2 level: *** set by the device administrator, can modify the parameter range: parameter level >=2

3 level: 3333 set by the machine operator, can modify the parameter range: parameter level >= 3;

4 level: *** not be input, can modify the parameter range: parameter level >=4.

The parameter level is referred to the parameter lists in APPENDIX.

4.6.1.2 Entering Operation Level

Entering the operation level is as follows:

① Enter the parameter password input window;

② Input the operation password(the system adds one * when one number is input to the password);

③ Press ENTER after the input is completed, i.e. the system enters the operation level corresponding password.

4.6.1.3 Parameter Management

The parameter management includes the parameter display, the parameter privilege, initializing, solidifying, draw, sending and receiving the parameter according to the privilege.

【Parameter display】

Parameter color definitions:

The parameter permitted to modify by the system is displayed in yellow and the forbidden is displayed in white in the current privilege.

In the parameter window, some operation option is related to the privilege; press hp1 and the forbidden functions are displayed in grey.

Prompt message display:

The prompt information of the parameter value can be opened or closed by setting the parameter P415_d7 when the parameter input is performed.

Other display:

When the input exceeds the modification parameter, the system prompts No modification authority in the parameter setting area.

In the parameter window, when the successfully modified parameter has a mark "*" before its parameter number, the system prompts the modification is completed successfully.

After the system executes some operations, the system displays the operation results and the successfully modified parameter has a mark "*" before its parameter number，prompting the modification having been completed successfully.

【Parameter privilege】

For the different privilege, the parameter which can be modified is displayed in yellow; the forbidden is displayed in white. The parameter update (using USB to transmit the parameters) is to modify the parameter data in the current privilege.

Privilege modification

The icon ahead of the parameter indicates the parameter is alterable under the current password level privilege; indicates the parameter is unalterable.

Privilege modification

The privilege modification is controlled by the password, and the password input is executed

when the system enters the parameter password input window. Whether the password can be memorized is controlled by **P416_d7**. **P416_d7**=0: it is not memorized, and the system enters the parameter window in other working modes and the parameter password input window is displayed as follows:

1) Modify **P416_d7**=0;

2) Press the menu key in any working modes except for PARAMETER working mode;

3) Press PARAMETER to enter the parameter password input window;

4) Directly input the operation privilege password;

5) Press ENTER after the input is completed, and the system enters the operation level corresponding to the password. (Set **P416_d7** to 1to memorize password)

【Parameter save】

The successfully modified parameters are automatically saved to the system, and all parameters are saved when the system exits from the parameter window (entering the window in the other working mode by press key). When the system is turned on every time, it reads the saved parameter data. When the saved data in read exceeds the range, it is rewritten to the minimum in the range and the system prompts it. The read parameter in disorder in power-on, the system prompts whether the previous solidified parameter is read; when the parameters have not been solidified, the system prompts to select the stepper/servo parameter to execute the parameter initialization and to save them to the system. The main differences between the stepper and servo parameter are the different motion parameter values.

4.6.2 Parameter Modification

The system parameter has been initialized before the factory delivery. The operator can modify and regulate correspondingly the parameters according to the actual conditions of the machine.

The system displays the selected parameter number in highlight after the parameter is selected.

4.6.2.1 Parameter Search

The parameter search is to search the required parameter as follows:

Method 1:

- Select the required parameter in the parameter window: **M-reference parameter, X-motion parameter, Z-transmission parameter, S-auxiliary parameter, T-bit parameter, U-interface parameter, W-variable initial value**.

  Example: select **M-reference parameter**, press **M** to enter the **reference parameter** window.

- Press , or , to move the highlight display to the parameter number which needs to search; the bit parameter: press , to left or right move the cursor to select the different bit, and the bit definition of the selected bit changes.

Method 2:

Directly position to the required parameter as follows:

Press P and input the required parameter number which needs to search, and then press ENTER. The system displays the parameter in highlight. Example: for searching P208, firstly input P, and then input 2 0 8, press ENTER and the parameter P208 has been found.

4.6.2.2 Parameter Modification

Modifying the parameter as follows:

① Search the parameter to modify it according to the above parameter search method.

② Press INPUT, and input the parameter data; or directly input the parameter data.

③ Press to delete the mistaken data and input again the correct data.

④ Press ENTER to confirm the operation.

【Note】

1) When the input data exceeds the parameter limit range, the input data is invalid and the parameter content does not change.

2) After the data is input, ESC is pressed and the input data is invalid.

3) Bit parameter input is as follows:

① After the required parameter which needs to be modified is selected, the operator can modify the parameter bit by the left/right direction key (prompt the current bit explanation at the bottom screen).

② Single bit modification: directly input the data which needs to modify (“0” or “1”: pressing other keys to input are invalid).

③ Modifying all bit: it is the same that of general parameter manual setting from left to right to input. For example: input 11, the parameter after the operator presses ENTER is modified into: 00000011; input 11000000, the parameter after the operator presses ENTER is modified into: 11000000;

4.6.3 Parameter Function Key Prompt hp1

Press hp1 in parameter window to display the direction of parameter function key prompts, and then perform operation by pressing keys according to the prompts.

The operator can perform the communication, extraction, solidifying, and upgrade the system software, and update the whole memory according to the password level; select “1” for stepper parameter initialization, select “2” for servo parameter initialization, select “3” for machine tool builder parameter extraction; select “K” to solidify the parameter; select “P” to search the parameter; select “F” to upgrade the software (through USB mode); select “D” to update the whole memory. Select the USB communication of parameter file “hp2” (File communication by USB).

When the system executes the data solidifying and extraction, it must not be turned off, and the

operator cannot execute the other operations before the operation is performed. The data solidifying and draw do not influence the part programs in the system.

4.6.3.1 Parameter Communication and Standard Format

According to the requirement to select the parameter transmission direction, the communication has two kinds: parameter sending and receiving. The parameter receiving includes: **USB→CNC,**; the parameter sending includes: **CNC→USB**

Parameter sending: (operation level: all level)

The operator in all level can send the parameter to U disc.

Parameter receiving: (operation level: machine manufacturer, device administrator, machine operator).

The operator with more than 3-level can receive the parameters from U disc However, it is only valid for those parameters which can be modified in the corresponding level.

1. USB operation：USB→CNC, CNC→USB

Firstly, control the hp1, then the hp2 to transmit the data by USB communication mode. The operator selects the transmission direction based upon their requirements.

When the parameter transmission is executed by U disc, the U disc root catalog needs to create one file “C001PAR”, the parameter sending and receiving are executed in the file. The file name format: “PAR”+file number (3-bit) +“.TXT”

2. Standard format of TXT parameter file in U disc：(Refer to the format of the parameter file that the system outputs to U disc).

In the U disc, the operator can use TXT, LST text to edit the parameter file, but the file name and file content must be compiled according to the required standard format to correctly send to the system. Refer to the parameter file format of the system outputting as follows:

1) In PC, the operator should rename the parameter file name to TXT or LST suffix, such as “**PAR099.TXT**”; it is suggested the operator should use the TXT suffix to conveniently operate the parameter file on PC.

2) The home of TXT file content must be the parameter mark: “CNC_GSKC001”; the item must exist.

3) The second line is the annotation, the front must have “//”; the item must exist.

4) The third line is the parameter content. The parameter content must meet its standard format requirement.

Example: **P000：**00000.000 // Z program reference point

P000: it is the parameter number, its format is “P + number + ：” which is the parameter number. The three parts are indivisible to consist of the parameter, the parameter number is not correct when it lacks one. 00000.000 is the parameter content, “//” is the parameter annotation.

5) The file content can be some of all parameters.

6) Standard format of parameter file communication on PC:

TXT file format	Remark

CNC_GSKC001
//Reference coordinate；
P000： 00200.000 // Z program reference point
……
P026： 0 // Y zero offset
//motion parameter;
P100： 6000 // Z max. traverse speed
……
//Driving parameter;
……
P209： 1200 // spindle encoder lines
//miscellaneous function;
P300： 1000 // max. speed of M41 gear
……
P344：0
//Bit parameter;
P400： 00000000 // running setting
……
//interface parameter;
P512： 0 // feed device alarm check Dalm
……
//variable initial value;
P600： 0 // variable r01
……
	Mark for checking parameters, which cannot be omitted.
	
	The contents behind // is the comment, which can be omitted.
	
	PXXX parameter number, P000: the first parameter is necessary.
	
	0: the parameter content is necessary.
	PXXX: parameter number and colon ":" which cannot be separated.
	
	……: means other parameters and the table does not list all parameter. The parameter receiving can be some of all parameters.

4.6.3.2 Parameter Draw and Solidifying

The parameter seen in the window are saved to the system SRAM storage which has the power-down protection function; when the main board battery has problem, the parameter loses.

The system has the parameter initialization function; according to the differences of the matched motor drive unit, the system can execute the different initialization operation; the system matched with DA98 series drive unit should execute the servo initialization, the system matched with DY3 series drive unit should execute the stepper initialization. The main differences of the initial parameter of the servo and the stepper are X-motion parameter and others are the same. The differences of servo/stepper initialization value are referred to the motion parameter list in the appendix.

The initial parameter does not meet all machines and the machine manufacturer should modify the spindle, the tool post and other parameters according to the detailed configuration of the machine.

To avoid the parameter loss, the system should execute the solidifying command, i.e. the modified parameter is solidified to the system FLASH storage to backup, the FLASH storage without the battery has the permanent save function. When the current parameter loses, the system extracts the solidified parameter to recover it.

The system pops-up the dialog box of relative operations about the parameter draw, solidifying, and the operator executes the option operation according to the corresponding dialog box. The parameter draw command includes: stepper parameter initialization, servo parameter initialization and machine manufacturer parameter draw.

【Explanation】

1) Before executing the parameter solidifying operation, the system should check the corresponding parameter, and the system prompts the alarm message to require the

operator to modify the corresponding parameter when the system find the parameter problem exists; after the parameter passes the check, it is saved to the system FLSAH to solidify.

2) The parameter solidifying operation must be executed before the solidified parameter draw.

4.6.3.3 System Software Upgrade and Memory Update

The system software upgrade is to replace the new system software, i.e. the old software version is replaced by the new; the main aim is to perfect the system function.

The system update is to update and improve the system software, so as to make the system more stable. It does no harm to the system, but incorrect operation may result in the system update failure. If the update fails, the system cannot be started up, or even the system hardware will be damaged. Therefore, we suggest the user not upgrade the system software without authorization. It is recommended that the upgrade service be provided by our professional staff with our after-sale service center.

The system memory whole update is to replace the memory including the system software covering memory.

The result of the memory whole update failure is more serious. It is suggested that the operator cannot use personally and that our personnel provides the service.

The system upgrade and memory whole update mode: USB mode; which is performed only by the machine tool builder.

1) System software upgrade by USB:

After U disc is inserted, the system automatically performs the software upgrade according to the system prompt to execute the operation. When the system software uses USB mode to upgrade, U disc root needs creates one file “C001DATA”, the command sending and receiving must be in the file. The file name format: “DATA”+ file number (3-digit) +“.TXT”. file name range: 0~254.

When the memory whole update uses USB, U disc root catalog needs to create the file “C001MEMO”, the command sending and receiving are executed in the file. The file name format: “MEMO”+file number (3-digit) +“.TXT”. File number range: 0~254.

【Note】

When the system uses USB software upgrade; the operator presses ESC or RESET to return EDIT working mode to edit the current program once again after the upgrade is completed successfully, otherwise, the system alarms.

4.6.3.4 Functional Command Privilege

In PARAMETER window, some operation functional option is related to the privilege, the forbidden operation functional option is displayed in grey.

The followings are the functional option and password privilege:

Operation privilege / Operation option	1-level Machine manufacturer	2-level Device administrator	3-level Machine operator	4-level Not input password
Stepper, servo parameter initialization	★			
Extracting parameter of machine manufacturer	★	★	★	
Executing parameter solidifying operation	★	★	★	
Parameter received by USB	Based on parameter level	Based on parameter level	Based on parameter level	Based on parameter level
Parameter sent by USB	All	All	All	All
System software update and memory whole update	★	★		
"★" in the above table means the option uses the privilege; The blank means the option has no use privilege.				

4.6.4 Parameter Explanation

The parameters are described according to the functions and uses and their detailed definitions are the followings.

4.6.4.1 Reference Point, Software Limit Bit Parameter __ P000¯¯P020

The reference point parameters include all important coordinate positions of machine Z/X/Y, and the each axis motion is based on these positions.

【Z/X/Y program reference point】__reference coordinate parameter P000, P001, P002

This parameter is used to set the position of program reference point. In AUTO/MANUAL mode, the machine returns to this position after program reference point return. The program reference point can be modified by inputting command (in MANUAL mode) or executing G50 command (in AUTO mode). The coordinate value of program reference point is the coordinates of the machine is not influenced by the tool offset value.

【2nd, 3rd program reference point of Z/X/Y】__reference coordinate parameter P003, P004, P005, P006, P007, P008

The 2nd and 3rd program reference point is similar to program reference point; In AUTO mode, after a specified axis performs G30, it returns to 2nd or 3rd program reference point commanded by G30. For details, please see to 4.10 G30 — 2nd, 3rd Program Reference Point Return in PROGRAMMING.

【Positive/negative tool nose software limit of Z/X/Y】__reference coordinate parameter P009,

P010, P011, P012, P013, P014

The parameter is used to limit the motion range of tool nose coordinates. It confirms the max. positive/negative travel of tool post in Z, X, Y.

In JOG/AUTO working mode, the tool nose coordinates of Z, X, Y are more than or equal to the positive tool nose software limit value, the axes only executes the negative instead of the positive motion. Otherwise, the system alarms: positive tool nose/cutter soft limit alarm. The negative motion is also so.

【Positive/negative machine software limit of Z/X/Y】__reference coordinate parameter P015, P016, P017, P018, P019, P020

The parameter is used to limiting the motion range of machine coordinates. It confirms the max. negative travel of machine in Z, X, Y.

In JOG /AUTO working mode, the machine coordinates of Z, X, Y are more than or equal to the positive mechanical software limit value, the axes only executes the negative instead of the positive motion. Otherwise, the system alarms: positive mechanical software limit alarm. The negative motion is also so.

4.6.4.2 Parameters Related to Zero Return Function __ P021～P026, P109～P111, **P123～P125,** P406～P407

[Machine zero coordinates of Z/X/Y]__reference coordinate parameter P021, P022, P023

The parameter confirms the coordinates of machine zero position.

When the machine installs the machine zero check device and **P407_d1=0**, after the operator executes the “Machine zero return (or G28) in JOG /AUTO working mode and the system has detected the “zero signal”, the system changes the current machine coordinate to the setting value of P021/P022P023.

[Zero offset value of Z/X/Y]__reference coordinate parameter P024, P025, P026

When the system uses the servo motor, checking the deceleration signal and the zero turn signal of the motor mask are taken the zero return check; The deceleration signal installed on the coordinate axis coincides with the zero turn signal, which can influence the zero return precision; the operator should set the offset value to 2mm and avoid the 2mm to execute the check.

[Zero return low-speed of Z/X/Y]__ motion parameter P109, P110, P111

Zero-point low-speed return speed is the movement speed of coordinate axis when the zero-point signal detection being performed in the process of machine zero return. No matter this speed is higher or lower than the initial speed, the lower one should be used; do not modified this speed after it is set; otherwise, the precision may be affected.

[Zero top rapid limit of Z/X/Y] __ Movement parameter P123, P124, P125

During the system performs the zero, and it returns to zero based upon G0 speed. If G0 speed is more than the zero top one, and the zero return speed is regarded as the most rapid limit.

Parameter **P123, P124**and**P125** are determined the coordinate axis top traverse speed limit during the Z, X and Y axes performs in the machine zero function return. As well, the actual zero

return speed of the Z, X and Y axes are controlled by the rapid feedrate.

The actual rapid zero return speed along with Z axis =P123×rapid feedrate (Unit: mm/min)

The actual rapid zero return speed along with X axis =P124×rapid feedrate (Unit: mm/min)

The actual rapid zero return speed along with Y axis =P125×rapid feedrate (Unit: mm/min)

【Zero setting 1】__ bit parameter P406(password class：1)

d7	d6	d 5	d 4	d 3	d2	reserved	D0

d7__ Z has or no deceleration signal

d6__ X has or no deceleration signal

d5__ Y has or no deceleration signal

0： None. // it is set to 0 when the deceleration switch and block are not installed on the machine;

1： Have. // it is set to 1 when the deceleration switch and block are installed on the machine.

d4__ Z has or no zero signal

d3__ X has or no zero signal

d2__ Y has or no zero signal

0： None. // it is set to 0 when the machine zero switch and block are not installed on the machine;

1： Have. // it is set to 1 when the machine zero switch and block are installed on the machine.

// The system has four kinds of zero return method and the detailed connection methods are referred to CONNECTION as follows:

Zero return method	Deceleration signal	Zero signal	Remark
1	Have	Have	**Use the deceleration signal and zero return signal to return the zero coordinates of machine zero**
2	Have	None	**Use the deceleration signal to return to machine zero**
3	None	Have	**Use zero signal to return machine zero**
4	None	None	**Return to zero coordinates set by the parameter(no machine zero)**

D0__ Without zero, zero return key is

0: Disabled. //When the system has no machine zero, it is invalid to press “Machine Zero Return key” on the panel, but program G28 zero return is still valid.

1: Enabled. //When the system has no machine zero, it is valid to press “Machine Zero Return key” on the panel. The system returns to the machine zero according to the 4th zero return mode.

//When there is neither a machine zero switch and block nor a deceleration switch and block installed on the machine, the system returns to the machine zero (i.e. returning to the zero coordinates set by parameters when there is no machine zero) according to the 4th zero return mode.

【Zero setting 2】__bit parameter P407(password class：1)

d7	d6	d 5	d 4	d 3	d2	d1	reserved

d7__ Z zero return direction

d6__ X zero return direction

d5__ Y zero return direction

0： Positive. // when the deceleration switch is installed on the positive end of the coordinate axis, it is set to 0, and the system positively moves the check zero;

1： Negative. // when the deceleration switch is installed on the negative end of the coordinate axis, it is set to 1, and the system negatively moves the check zero;

d4__ Z deceleration signal level

d3__ X deceleration signal level

d2__ Y deceleration signal level

0：LOW. // The system decelerates to stop after it has checked the deceleration signal to be LOW in the coordinate axis zero return.

1：HIGH. // The system decelerates to stop after it has checked the deceleration signal to be HIGH in the coordinate axis zero return.

d1__after the machine zero return is executed, whether the machine coordinates are modified.

0： Modifying machine coordinates // generally, it is set to 0; the system modifies the machine coordinates after it executes the zero return.

1： Not modifying machine coordinates // in debugging zero return function, it is set to 1 to learn the zero return precision.

4.6.4.3 Traverse Speed, Acceleration Time Parameter __P100～P108, P112～P119

Regulating P100～P116 to make the system meet the motor with the different type and the machine with the different load to improve the machining efficiency.

Example: regulate the parameter value to the low to avoid the step-out when the system uses the stepper motor; properly increase the parameter value when the system uses the servo motor.

The detailed is referred to **OPERATION, 4.6.5.2 Appendix**, the difference between the stepper initial value and servo initial value in the motion parameter list.

【Rapid traverse speed limit of Z/X/Y】__motion parameter P100, P101, P102

P100, P101, P102 confirm the rapid traverse speed of Z, X, Y in manual rapid and G00, the actual rapid traverse speed in Z, X, Y is controlled by the rapid override.

Z(X, Y) actual rapid speed =P100 (P101, P102)x rapid override (unit: mm/min)

【Lowest initial speed of Z/X/Y】__motion parameter P103, P104, P105

P103, P104, P105 defines the lowest initial speed of Z, X, Y in MANUAL or AUTO mode. When Z, X, Y speed are lower than the setting value of **P103, P104, P105**, Z, X, Y has no speed rising and reducing process. The user should adjust the value properly based on the actual machine tool load. (Unit: mm/min)

【Rapid feed acceleration/deceleration time (ms)of Z/X/Y 】__ motion parameter P106, P107, P108

P106, P107 and P108 defines the time of Z, X, Y linearly increasing speed from 0 to 15m/min in MANUAL or AUTO mode. The larger the**P106, P107 and P108** values are, the longest the

acceleration processes of Z, X, Y are. The operator should reduce **P106, P107 and P108** values to improve the machining efficiency based on meeting the load characteristics. (Unit: ms).

【Cutting feed initial speed】__ motion parameter P112

P112 feed initial speed. **P112** defines the initial speed of commands such as G01 in automatic machining process. When F value is smaller than the one set in P112, there is no speed rising and reducing process. (Unit: mm/min)

【Max. speed limit of cutting feed】__ motion parameter P113

P113 max. speed of cutting speed. **P113** defines the max. speed of commands such as G01 in automatic machining process. When the specified F speed is larger than the setting value of P113, there is no speed rising and reducing process. When thread cutting speed is more than **P113**，the system will alarm and stop thread cutting. (Unit: mm/min)

[Linear acceleration/deceleration time of cutting feed (ms)]__ Motion parameter P114

P114 feed linear acceleration/deceleration time;

P114defines the time spent from 0 speed to **15**m/min of commands such as G01 in automatic machining. (Unit: ms)

[Acceleration/deceleration time of thread cutting (ms)]__ Motion parameter P116

P116 defines the time spent from 0 speed to **15**m/min of thread cutting axis. The smaller P116 value is, the shorter the inconstant is. When **P116** is too small, the stepper motor may cause the step-out. (Unit: ms).

【G99 initial lines】__ motion parameterP118

When the parameter is not 0 and the system executes G99 every time, it checks the spindle encode lines till the lines reaches the value set by P118, i.e. the spindle rotates some angle to execute the other commands following G99.

When the parameter setting value is 0, the system does not check the encoder lines till it executes other commands following G99.

The line that the system detects is four times that of the spindle encoder.

E.g. If the lines of the encoder installed is 1200, the lines that the system detects changes between 0~4800 recurrently. This parameter should be set between 0～4800 as well, or the system keeps waiting because G99 command cannot be detected.

[Delay time when orientation is switched to cutting (ms)]__Motion parameter P119

If the last command is a rapid positioning command, and the current one is a cutting command, the delay time set in P119 is automatically inserted between these two commands; this parameter is used for preventing the taper streak on the workpiece due to the excessively fast command transition. If the parameter is too large, the machining efficiency may be affected. In general, set it between 0～100.

Example 1:

```
G00 U-50          ；Rapid positioning
G01 W-10 F100 ；The command following it is a cutting command, so the delay time set in P119 is
```

inserted before executing this command; which prevents taper streak on the front end.

Example 2：

G00 U-50 ； Rapid positioning

S1000

G01 W-10 F100 ； As it does not follow the rapid positioning command directly (another command separate them) , the delay time will not be inserted automatically.

4.6.4.4 Parameters Related to Transmission and Compensation__ P200～P209, P411

【Z/X/Y command pulse multiplication ratio】__transmission parameter P203, P205, P207

【Z/X/Y command pulse division coefficient】__ transmission parameter P204, P206, P208

P203 — Z multiplication: Z electronic gear numerator. (range: 1～9999999).

P204 — Z division: Z electronic gear denominator (range: 1～9999999).

P205 — X multiplication: X electronic gear numerator. (range: 1～9999999).

P206 — X division: X electronic gear denominator (range: 1～9999999).

P207 — Y multiplication: Y electronic gear numerator. (range: 1～9999999).

P208 — Y division: Y electronic gear denominator (range: 1～9999999).

【Note】

1) In setting the related parameters of the transmission, the ratio between the multiplication coefficient and division coefficient is 1/128 ～128, otherwise, the system alarms.

2) The ratio between the frequency-multiplication coefficient and the frequency-division is 1:1: Z/X/Y axes: the system outputs 1 pulse when it moves 0.001mm every time;

3) The ratio between the frequency-multiplication coefficient and the frequency-division is not 1:1: Z/X /Y axes: the system outputs the pulse in movement of 0.001mm: multiplication/division;

4) When the system max. pulse output frequency is 511pps/ms, the frequency cannot exceed the value, otherwise, the system alarms in movement. i.e. P100 (max. traverse speed) x multiplication/division should not be more than 30000mm/min.

【Spindle encoder lines】__ transmission parameter P209

The parameter defines the spindle encoder lines, it setting range: 100～5000. When the spindle is ON, on the initial page of DIAGNOSIS mode, if the setting value is not consistent with the spindle encoder lines, the diagnosis prompts in DIAGNOSIS working mode: Encoder lines are not consistent with the parameter. Unit: line/r.

【Z/X/Y backlash value】__ transmission parameter P200, P201, P202

The parameter defines the backlash value of Z, X, Y mechanical transmission. Unit: mm.

There are backlash clearances in the lead screw, the decelerator and other driving device, which cause the error in the repeated motion of tool post. To avoid the error, set P200, P201, P202 which make CNC system automatically compensate the error when the machine changes its moving direction.

Measurement method of mechanically-driven backlash (taking example of Z):

1) Select the proper feedrate in JOG working mode.

2) Install the dial indicator on the proper position of the machine, move the tool post to the probe of the dial indicator and set its pointer to zero.

3) Select STEP working mode with the step size 1.0 mm.

4) Press Z axis coordinate axis movement key to move the tool post to the dial indicator and make it point to zero when rotating one circle.

5) Press Z axis coordinate axis movement key to move reversely and the pointer of dial indicator turns around. The pointer cannot return to zero because of the backlash. At the moment, D-value between the pointed position of pointer and zero is Z backlash value.

【Notice】

1) Repeat the above-mentioned operations many times to gain the exact measurement value.

2) The measurement method of X backlash is the same.

3) Z, X, Y backlash compensation speed is to compensate with the speed set by **P411_d1**.

【Precision compensation】__bit parameter P411(password level：2)

Reserved	Reserved	Reserved	Reserved	Reserved	Reserved	d 1	Reserved

d1__backlash compensation mode

0： Low-speed. // Z/X/Y executes the backlash at the low speed; the low speed value is P103, P104, P105.

1： Fast. // Z/X/Y executes the backlash at the fast; the fast value is P100, P101, P102.

4.6.4.5 Parameters Related to Spindle, Cooling __ P300～P317, P326, P329, P341,P350, P351, P410

【Spindle configuration】__bit parameter P410(password level：1)

d7	d6	d 5	d 4	d 3	d 2	d1	reserved

d7__spindle controlling output

0： Level. // the system level output controls the spindle (M03/M04/M05), cooling (M08/M09).

1： Pulse. // the system pulse output controls the spindle (M03/M04/M05), cooling (M08/M09). P326 is the pulse width.

d6__spindle S control

0： Gear shifting. // the spindle is the gear shifting spindle, and the system uses S to execute the gear shifting; forbid M41～M44.

1： Frequency. // the spindle is the frequency, S is the spindle speed, M41～M44 gear.

// For details, see Section 4.4.3.2“Spindle S Command—Gear Shifting Control” and Section 4.4.3.3“Spindle S command—Speed Control”.

d5__S gear shifting output

0： Direct. // the control signal of spindle gear is to directly output S01～S04, forbid S05～S15.

1： Encoder. // the control signal of spindle gear is the encode output S00～S15.

// the parameter and P310 (spindle gear control covered lines) are used together. When the setting is the direct output, each control line controls one gear, and the all gear quantity and P310 quantity are consistent; when the encode output is according to the used control line quantity, the controlled total gear quantity is 2^{P310}; max. setting 4-channel control line output is taken as the gear control signal.

d4__switching the spindle and Y

0： Do not switch. // the spindle does not work in the position control mode, forbid M47/M48.

1： Switch. // the spindle switches between the position control mode and the speed control mode, use M47/M48.

d3__spindle brake signal output

0： Use. // output the brake signal MSP in executing M5.

1：Do not use. // do not output the brake signal MSP in executing M5, MSP signal interface is used to other interfaces control.

d2__spindle CCW signal output

0： Use. // output the spindle rotation (CCW) signal M04

1： Do not use. // forbid outputting the spindle (CCW) signal M04

d1__YM will be cleared when M47 performs

0: Cleared. //The machine coordinate YM will be cleared when performing the M47.

1: No clear. //The machine coordinate YM will not be cleared when performing the M47.

【max. speed of M41, M42, M43, M44】__ auxiliary parameter P300, P301, P302, P303

The parameter is max. spindle speed of M41, M42, M43, M44. When the system uses the converter controlling the spindle and the spindle gear is M41, M42, M43, M44 and the system outputs 10V analog voltage, it corresponds to the max. speed of the machine. P300 P301 P302 P303 are invalid when the spindle multi-gear switching controls the spindle. (Unit: r/min)

【Lowest speed of spindle with constant surface】__ auxiliary parameter P304

The parameter defines the lowest speed in the constant surface cutting, the calculation formula according to the constant surface cutting speed:

Surface speed=spindle speed *|X|*π/1000(X unit：mm，spindle speed unit: r/min)

X in the formula is the tool nose coordinate value on X axis. When X is less than some value and the calculated spindle speed is less than P304 value, the spindle speed keeps P304 speed to execute the machining.

【Max. spindle speed with constant surface speed】__ auxiliary parameter P305

The parameter defines the max. speed in the constant surface cutting, the calculation formula according to the constant surface cutting speed:

Surface speed=spindle speed *|X|*π/1000(X unit：mm，spindle speed unit: r/min)

X in the formula is the tool nose coordinate value on X axis. When X is more than some value and the calculated spindle speed is more than P305 value, the spindle speed keeps P305 speed to execute the machining.

【Thread smooth speed borderline】__ auxiliary parameter P306

The system has two methods in machining thread: high-speed and low-speed machining. P306 is the borderline of the two machining speed (unit: r/min). Generally, P306 should be more than 100.

P306=300, before the system executes the thread machining, it should judge according to the current actual checked spindle speed. When the actual speed is more than 300, the system uses the high-speed machining; otherwise, it uses the low-speed machining mode.

In high-speed machining, the system has the strongest following to the spindle speed wave, but the stability of the motor running reduces; when the spindle speed wave is strong, there is the wave on the machining surface and the stepper motor causes the step-out.

In low-speed machining, the system has the poorer following to the spindle speed wave, but the stability of the motor running increases; when the spindle speed wave is strong, the wave on the machining surface is worse and the stepper motor does not cause easily the step-out.

【Thread spindle wave alarm】__ auxiliary parameter P307

When the spindle speed wave exceeds P307 in thread cutting, the system alarms after the current thread block machine is completed and dwell the machining workpiece; the next step of the machining programming can be consecutively performed if the CYCLE START is controlled.

The parameter is enabled to G33, G32, G34 commands. (Unit: r/min)

【Spindle JOG time (ms)】__ auxiliary parameter P308

In spindle JOG mode, it is the time for starting the spindle, then the spindle stops automatically after the time is up. When parameter P308 is 0, the spindle JOG function is disabled. (Unit: ms)

【Speed in spindle JOG】__ auxiliary parameter P309

In spindle speed, after pressing the spindle start key when the spindle is in JOG state. (Unit: mm/min)

P309=0: the JOG output speed is the same that of M03/M04.

【Used lines for the spindle gear control】__ auxiliary parameter P310

The parameter limits the used output controlled line quantity of spindle gear control, up to 4 control lines. When the control line quantity is less than 4, the system only uses the low-bit control line, and the used high-big control line is used to others.

The used line quantity=0, the system does not output.

The used line quantity=1, only **S01** outputs.

The used line quantity=2, only S02, S01 output.

The used line quantity=3, only S03, S02, S01 output.

The used line quantity=4: S04, S03, S02, S01output.

【Frequency spindle gear shifting time 1, 2(ms) 】__ auxiliary parameter P311, P312;

When the spindle is the frequency, M41~M44 spindle gear shifting time is shown as follows. (Unit: ms). Refer to **OPERATION 4.4 JOG working mode**.

【Spindle gear shifting interval time(ms) 】__ auxiliary parameter P313

This parameter set the interval from original gear signal being cancelled to new gear signal being output. (Unit: ms)

【Output voltage in spindle gear shifting(mV) 】__ auxiliary parameter P314

Output voltage in the spindle gear shifting. (Unit: mV). See Section 4.4.3.3 Spindle S Command—Rotation Speed Control in Part Ⅰ Operation for details.

【Brake delaying time in spindle stop(ms) 】__ auxiliary parameter P315

The parameter defines the delay time from sending the spindle stop signal to the spindle brake signal. (Unit: ms)

【Output time in spindle brake(ms) 】__ auxiliary parameter P316

The parameter means the output time of brake signal, and the time is set from the spindle motor brake starting to its exact stop

【Max. speed limit of spindle】__ auxiliary parameter P317

This parameter limits the spindle maximum speed. (Unit: r/min)

If the S speed set by the program command is greater than the value given by P317, the spindle speed is clamped to the one set in P317.

【Pulse time controlled by the spindle(ms) 】__ auxiliary parameter P326

The parameter defines the durable time of the pulse signal when the system executing the spindle (M03/M04/M05) and cooling (M08/M09) output are the pulse control mode. (Unit: ms)

【M respond check time(ms) 】__ auxiliary parameter P329

The parameter defines the upper limit of M respond check time, the system alarms when it has not checked the M respond signal in the time. (Unit: ms). For example: execute M10 in JOG working mode to wait P329 time, and the system alarms Alarm for chuck clamping responding check overtime when it has not checked the chuck clamping responding signal (**P409_d5**=1).

【Cutting enabled when rotation speed reach certain percentage】__auxiliary parameter P341

If the cutting is started when the spindle is just turned ON or gear shift is just finished, as the preset rotation speed is not reached, the cutting will affect the tool life and workpiece.

This parameter is used to identify spindle rotation speed reached or not. For example, P341=80 means only the spindle actual speed can be cut (80%～120%) between the 100%±20% set by program; If the program is S2000, the actual speed detected by system can be cut, which will enter 1600～2400, otherwise, you should keep waiting.

When P341=0, this function is disabled. When P341 is not zero, in MANUAL or AUTO mode, the actual speed is detected automatically and displayed on the screen; if the spindle speed Sxxxx is displayed in yellow, it means the preset range is not reached; if green, it means the preset range is reached.

Usage: This parameter is valid when inverter spindle or gear-shift spindle is used.

When variable-frequency spindle is used, the system compares the programmed speed S with the actual speed.

When gear-shift spindle is used, as S represents the gear, S1~S4 gear spindle rotation speed corresponds to the setting value of P300~P303; if the gear is S1, the system compares the setting value of P300 with the actual speed. P341 is invalid to other gears, so the cutting will not be affected by it.

This parameter is valid only for cutting command such as G01, G33 etc. When cutting

commands are executed continuously, only the first one is detected. If the spindle rotation speed is displayed in yellow, and the cutting is not performed, it means the system is waiting for the spindle to reach the preset speed.

[Spindle Clamping/Releasing Time (ms)] __Miscellaneous parameter P350

Spindle clamping/releasing time. Parameter range: 0~9999.

[Servo OFF Delay Time (ms)] __ Miscellaneous parameter P351

Servo OFF delay time. Parameter range: 0~9999

4.6.4.6 Parameters Related to Tool Post __ P318～P319

【Tool post type】__ auxiliary parameterP318(max. value is 9)

The parameter is defined by the tool post type installed on the machine, and the system executes the tool change mode according to the parameter.

P318=0~8，the line-up tool post installed on the machine does not cover any input/output.

P318=9，use M60 customized command tool change, refer to examples in **CONNECTION 3.4 Tool Change Function and Connection.**

【Max. tool selection number】__ auxiliary parameter P319

The parameter defines max. tool selection number of the tool post. The standard configuration of the system CNC is 8 tool selection electronic tool post. The tool selection signal can extend to 12～16 tool selections tool post according to the special encode input. Refer to **CONNECTION, 3.4 Tool Change Control Function and Connection**.

4.6.4.7 Parameters Related to Chuck and Tailstock __ P327～P328, P409

【Chuck control pulse time(ms) 】__auxiliary parameter P327

When the chuck is pulse control mode, the parameter defines the durable time of the chuck execution command (M10/M11) outputting pulse signal. (Unit: ms)

【Tailstock control pulse time(ms) 】__ auxiliary parameter P328

When the tailstock is the pulse control mode, the parameter defines the durable time of tailstock execution command (M78/M79) outputting pulse signal. (Unit: ms)

【Chuck tailstock】__bit parameter bit parameter P409(password level password level：1)

d7	d6	d 5	d 4	d 3	d2	d1	d0

d7__chuck control function

0： Valid. // the system has the hydraulic chuck control function.

1： Invalid. // the chuck control function is invalid, forbids M10/M11.

d6__chuck clamping mode

0： Outer. // the hydraulic chuck is outer mode.

1： Inner. // the hydraulic chuck is inner mode. Executing M10/M11 is referred to **OPERATION, JOG Working Mode**.

d5__chuck respond check

0： Do not check..

1： Check. // when the system needs the respond check, input interface RM10 and RM11 are separately taken the clamping and releasing in-position signal input.

d4__tailstock control function

0： Valid. // the system has the hydraulic tailstock control function.

1： Invalid. // the system forbids M78/M79 without the hydraulic tailstock control function.

d3__chuck control output

0： Level. // control M10/M11 output in level mode.

1： Pulse. // the hold time of M10, M11 in pulse control mode is determined by P327.

d2__tailstock control output

0： Level. // control M78/M79 output in level mode.

1： Pulse. // the hold time of M78, M79 in pulse control mode is determined by P328.

d1__hydraulic chuck foot switch input

0：Do. // besides using the command to control the hydraulic chuck, the operator can use the external foot switch to control the hydraulic chuck.

1： Do not.

d0__hydraulic tailstock foot switch input

0：Do. // besides using the command to control the hydraulic tailstock, the operator can use the external foot switch to control the hydraulic tailstock.

1：Do not.

4.6.4.8 Run and Efficiency Bit Parameter __ P400～P401, P420

【Run setting】**__bit parameter P400 (password level：3)**

Reserved	d 6	d 5	d 4	d 3	Reserved	d 1	d0

d6__spindle gear memory

0： Invalid. // the spindle gear M41～M44 cannot be memorized in JOG and AUTO working mode, it is M41 after power-on again.

1： Valid. // the spindle gear M41～M44 can be memorized in JOG and AUTO working mode, it is the previous gear before last power-off after power-on again.

d5__spindle in AUTO working mode

0： Controlled by the key. // In AUTO mode no program running state, pressing the spindle CW, spindle stop key is valid.

1： Not be controlled by the key. // In AUTO mode, pressing the spindle CW, spindle stop key is invalid.

d4__MPG 0.1mm step width

0： Valid. // select the big override (x0.1mm gear) to rotate the MPG in JOG working mode.

1： Invalid. // cannot select the big override (x0.1mm gear) to rotate the MPG in JOG working mode.

d3__rapid position mode

0： Alone traverse. // each axis rapidly positions separately in G00.

1： Interpolation traverse. // rapidly position in the proportion mode in G00.

d1__MPG axis selection

0： Keyboard axis selection. // keyboard axis selection is valid.

1： External axis selection. // the external MPG control knob is valid and the keyboard axis selection key is invalid.

// In JOG working mode, press MPG to switch MPG mode and the indicator lights and the system displays states of the external MPG control knob, the axis selection knob and the movement knob. The coordinates of the selected coordinate axis is displayed in highlight state.

d0__Feedrate selection

0: Keyboard enabled. //The keyboard feedrate button is enabled in the Manual/Auto mode. The system initialization is set to 0.

1: External enabled. //The external feedrate is enabled in the Manual/Auto mode.

【Efficiency setting 】__bit parameter P401(password level：3)

d7	d6	d 5	d 4	d 3	d 2	reserved	d0

d7__TMS command in DRY RUN working mode

0： Execution. // in DRY RUN working mode, output and check the signal in executing the miscellaneous command, which is the same in the normal machining.

1： Skip. // in DRY RUN working mode, do not output and check the signal in executing the miscellaneous command.

d6__feed command in DRY RUN

0： Constant speed. // in DRY RUN working mode, the execution speed of feed command is set by the program, which is the same in the normal machining.

1： Acceleration. // in DRY RUN working mode, the execution speed of feed command is not controlled by the program and the max. speed (P113) of cutting feed demonstrates the program path.

d5__short linear high-speed

0： High-speed connection. // the short linear connection transition uses the high-speed connection.

1： Do not use. // the short linear connection transition does not use the high-speed connection.

// **P401_d5**=0: for the short movement, the continuous short linearity (without blank line, annotation line or other block between blocks) with rapid execution, CNC uses the prospective control mode to realize the best speed connection transition, max. preread blocks reach 80.

d4__cutting command

0： Continuous smooth transition.

1： Decelerate to zero.

// In G01, G02, G03 cutting feed, when bit parameter **P401_d4**=0, the system uses the post acceleration/deceleration processing, and forms an arc transition at the path intersection point of the neighboring two cutting feed blocks, which cannot exactly position the intersection of the two paths and there is the contour error between the actual path and the programmed path. To avoid the contour error, the operator can insert G04 (delay command) between two blocks or set **P401_d4** to 1. At the moment, when the previous block moves to the end point of the block, it decelerates to zero and exactly positions to the end point of the block and then executes the next cutting feed block; in the mode, because each block accelerates from the initial speed, decelerates to zero in the end point to execute the next, the program run time increases and the machining efficiency reduces.

d3__multi-command execution sequence in the same block

0： Substep

1： Synchronism. / When there are many commands are shared with one block, which is called the multi-command in one block. These commands are almost executed and the system executes the next block till all are completed.

// **P401_d3**=0: the commands are executed in substep; the execution sequence: after one functional command is executed, the system executes the next one till all are completed.

// **P401_d3**=1: synchronous execution; in the mode, the command which are not relative each other are almost executed, do not follow the substep execution sequence till each is completed. The system automatically arranges the execution sequence for M with the interlock relationship.

d2__rapid skip block execution function

0： Close.

1： Open. // **P401_d3**=1: **P401_d2**=1 is valid. **P401_d2**=1: the system permits the skip block execution. In the mode, after the axis motion command of current block is executed(the system does not wait for M, S, T having been done), it rapidly skips to the next block to execute.

d0__Cutting efficiency

0：Normal // The cutting feedrate is the normal value of command F.

1：Optimized // Improve the efficiency by 5%; it applicable to situations that F command is not modified while efficiency is to be improve.

[Efficiency setting 2] __Bit parameter P420 (Password level: 3)

Reserved	Reserved	Reserved	Reserved	d 3	Reserved	d 1	Reserved

d3_Program nesting/built-in cycle

0: View. //Window appears by Upward direction key during the Auto machining components,

and the program nesting and cycle can be viewed.

1: Do not view.

d1__Perform the variable calculation

0: Normal/Constant.

1: Acceleration. //The system will compiled to the notes page by the performed statement command from compile when the parameter setting is regarded as the variable calculation acceleration. In this case, the operation time can be increased and the working efficiency can be enhanced.

4.6.4.9 Relationship Between Path and Run, Efficiency Parameter

1) **P401_ d4=0:** the cutting command executes the continuous and smooth transition and the post acceleration/deceleration function is valid:

G01 processing:：

P401_ d5=0: G01 has the front acceleration/deceleration function, G01 and G01 perform the connection transition with the best speed; and the system executes the post acceleration/deceleration processing; when the cutting speed is high, the linearity path is precise and the transition point is smooth. At the moment, the post acceleration/deceleration function does not greatly influence the path precision.

P401_ d5=1: G01 does not use the high-speed connection and execute the front acceleration/deceleration processing, and directly executes the post acceleration/deceleration processing.

2) **P401_ d4=1**: the cutting command decelerates to zero; the post acceleration/deceleration function is invalid:

G01 processing:

P401_ d5=0: G01 has the front acceleration/deceleration function, G01 and G01 perform the connection transition with the best speed; all G01 moves to the end point of the block and decelerates to zero, exactly positions the end point of the block; when the cutting speed is high, the linearity path is precise and the transition point is smooth. **P401_ d5=1**:G01 has the front acceleration/deceleration processing, decelerates to zero after each cutting command is executed.

4.6.4.10 Safety and Debugging Bit Parameter __ P402～P404, P419

【Safety setting 1】__bit parameter P402(password level：3)

d7	d6	d 5	d 4	d 3	d 2	d 1	d0

d7__hardware limit alarm

// the parameter specifies its using mode when the system has checked the hardware limit alarm signal in the axis motion process.

0： Deceleration. // when it is set 0, it meets the hardware limit alarm, the motion axis

decelerates to stop, and the coordinates are consistent with the actual position;

1： ESP. // when it is set 1, it meets the hardware limit alarm, the motion axis decelerates to stop, and the coordinates are not consistent with the actual position;

d6__spindle abnormal stop rotation check

0： Check. // the system stops the feed, closes the spindle and alarms in cutting.

1： Do not check.

// In JOG /AUTO working mode, after the spindle starts, the system automatically check the spindle speed, it prompts “spindle rotation abnormality” when the spindle starts abnormally. When the system stops rotating accidently in cutting feed, the system stops the feed, interrupts the program execution, and closes the spindle and alarms.

// When the spindle works in the low speed (less than 1 r/min), the system alarms, at the moment, the parameter should be set to “1”. For the frequency spindle, when the programming speed is less than S0, the system does not check and alarm.

d5__Interlock between chuck and spindle

0： Interlock. //when the chuck and the spindle are interlocked, the chuck cannot be controlled when the spindle is rotating; the chuck releases but the spindle cannot be started.

1： Releasing interlock.//when the chuck is not interlock with the spindle, starting the spindle is not influenced by the chuck state, controlling the chuck is not influenced by the spindle state.

d4__Chuck respond signal consecutive check

0： Do not alarm.

1： Alarm. // the bit means whether the system real-time check the chuck state, and alarms when the chuck releases. The control bit is valid when the chuck has the respond signal. (**P409_ d5=1**)

d3__Interlock between tailstock and spindle

0： Interlock. // /when the tailstock and the spindle are interlock, the chuck cannot be controlled when the spindle is rotating; the tailstock releases but the spindle cannot be started.

1： Releasing interlock. // when the tailstock is not interlock with the spindle, starting the spindle is not influenced by the tailstock state, controlling the tailstock is not influenced by the spindle state.

d2__Tailstock respond signal consecutive check

0： Do not alarm.

1： Alarm. // the bit means whether the system real-time check the tailstock state, and alarms when the tailstock releases. The control bit is valid when the tailstock has the respond signal. (Setting P519、P520)

d1__Automatic MPG control

0： Invalid. // using MPG to control rapid/feedrate override is invalid in AUTO working

mode.

1： Valid. // using MPG to control rapid/feedrate override is valid in AUTO working mode.

d0__Automatic MPG mode

0： Mode 1. // use the mode 1 to control MPG rapid/feedrate override.

1： Mode 2. // use the mode 2 to control MPG rapid/feedrate override. Mode 1 and 2 are referred to **OPERATION, Section 4.5 AUTO Working Mode**.

【Safety setting 2】__bit parameter P403(password level：3)

Reserved	Reserved	Reserved	Reserved	d3	d 2	d 1	d 0

d3__M output during ESP

0: Only spindle and cooling are turned off. // When the emergency stop alarm is valid, the system only turns off the signals of the spindle, coolant and lubricant.

1: All are turned off. // When the emergency stop alarm is valid, the system turns off all the auxiliary function output signals.

d2__Spindle and coolant in pressing reset key.

0： Close. //after the reset key is pressed, the system closes M03, M04, M08, M32 output signal.

1： Do not close. // after the reset key is pressed, the system does not close M03, M04, M08, M32 output signal.

d1__M function alarm

0： Terminating program. //program terminates when **M10 M11 M78 M79 Txx** alarm.

1： Prompt selection. //the system prompts whether the operator tries again when **M10 M11 M78 M79 Txx functions alarm.**

// When the system meets **P401_d3=0** (substep execution for many commands in one block) and **P403_d1=1** (prompt in M function alarm), the trial again function is valid. The system alarms and terminates the machining programs when the relative commands have not been completed.

The trail again commands include: T, chuck control (M10/M11) and tailstock control (M78//M79).

When the system executes these commands, it has not checked the corresponding valid input signals, and has not completed the command operations in the limit time, the system prompts whether the operator tries again. The system is in pause state when it prompts the trial message; at the moment, the operator should check the relative input signal and execute the troubleshooting.

After the failure is resolved, the operator presses R key execute again the command which is just now failure. After the re-execution is completed, the system is in pause state, and the operator presses “CYCLE START” to continuously the machine. When the re-execution is not correction, the operator should press “ESC” to exit the machining program.

d0__wait for stable speed before machining the thread

0:do not wait // whether the system checks the spindle speed is stable before machining thread.

1: wait // the system automatically checks whether the spindle speed is stable before

machining thread, and waits the stable speed to machine thread.

// The parameter is valid to G33, G32, G34.

// The system checks the spindle speed instantaneously; when the spindle raises speed or reduces speed, the system waits for the process to end and then machines the thread. When the system executes immediately the thread command after changing the speed, maybe the checking spindle speed function is invalid.

Example: in S1000 stable state, the system executes S200, and immediately executes G33, the system immediately executes the program for check speed being stable, at the moment, the system judges by mistake in S1000 stable state it is “having been stability”; because there is the time difference between the system command and the spindle speed. It should suggest that the operator should delay 0.3s between S200 and G33.

When the spindle with the low speed (below 10r/min) executes the thread cutting, the system cannot have checked the spindle speed stability for a long time; at the moment, the parameter should be set to “0” to cancel the function.

【Safety setting 3】__Bit parameter 419（Password level: 3）

d7	d6	d5	d4	d3	d2	d1	d0

d7__ Whether turn OFF the spindle or not when drive unit alarm occurs:

0：No // When a drive unit alarm is detected, the spindle is not turned OFF.

1：Yes // When a drive unit alarm is detected, the spindle is turned OFF.

d6__ The M state change is:

0： Not prompted // The state change of spindle, chuck and tailstock is not prompted in the system.

1： Prompted // The state change of spindle, chuck and tailstock is prompted in the system.

// During AUTO machining, if you want to stop in the machining process, you need to turn OFF the spindle, and later turn ON it, then, press CYCLE START to continue. Sometimes you may forget to turn ON the spindle, which may cause accident. Set P419_d6 to 1 can prevent such thing from happening. When P419_d6=1, in the initial state of single block or stop, the state of spindle (M3/M4/M5), chuck (M10/M11) and tailstock (M78/M79) are stored in the system. Press CYCLE START, the system will first identify whether the state is the same with the previous one. If not, a pop-up window is displayed prompting the changed contents. You can press Y to continue or press N to exit.

d5__Monitoring closes when ends

0: Close. //When it is set to 0, the monitoring close is viewed when opening the window after the program operation is ended.

1: Do not close. //When it is set to 1, the monitoring is still performed when opening the window after the program operation is ended.

d4__Monitoring closes when resetting

0: Close. //When it is set to 0, the monitoring close is viewed when opening the window after pressing the “resetting” button.

1: Do not close. //When it is set to 1, the monitoring is still performed when the window is opened after pressing the “resetting” button.

d3__Z axis allows G06 mode

0: Forbidden. //Z axis prohibits G06 movement mode.

1: Allow. //Z axis allows G06 movement mode.

d2__X axis allows G06 mode

0: Forbidden. // X axis prohibits G06 movement mode.

1: Allow. //X axis allows G06 movement mode.

d1__Y axis allows G06 mode

0: Forbidden. // Y axis prohibits G06 movement mode.

1: Allow. //Y axis allows G06 movement mode.

d0__G06 movement mode during resetting

0: G06 Stop. //Press resetting button, G06 stops feed when system performs the standard resetting function.

1: Without affection. //Press resetting button, the G06 does not affect while the system performs the standard resetting unction.

【Debugging setting】__bit parameter P404(password level：2)

d7	d6	d 5	d 4	d 3	d 2	d 1	d 0

The parameter is set for being convenient to the system being debugging, must be set to the valid state in power-on, otherwise, it cannot play a role in protection.

d7__emergency stop alarm

0： Check. // the external emergency stop signal function is valid.

1： Shield. // the external emergency stop signal function is invalid.

d6__hardware limit alarm

0： Check. // the hardware limit alarm function is valid.

1： Shield. // the hardware limit alarm function is invalid.

d5__drive unit alarm

0： Check. // the drive unit alarm function is valid.

1： Shield. // the drive unit alarm function is invalid.

d4__mechanical software limit alarm

0： Valid. // the mechanical software limit alarm function is valid.

1： Invalid. // the mechanical software limit alarm function is invalid.

d3__tool nose software limit alarm

0： Valid. // the tool nose software limit alarm function is valid.

1： Invalid. // the tool nose software limit alarm function is invalid.

d2__254 program limit

0： Forbidding motion command. // M50～M74 blocks forbid all G command (except for

G04) .

1： Permitting. // M50～M74 are the user customized commands. The block permits G commands according to the program format to compile programs.

d1__hardware limit alarm level

0：LOW. // the hardware limit alarm LOW is valid.

1： HIGH. // the hardware limit alarm HIGH is valid.

d0__254 program solidifying limit

0： Forbidding. // forbid modifying and solidifying No. 254 program, i.e. forbid modifying the customized command.

1： Permitting. // permit modifying and solidifying No. 254 program, i.e. permit modifying the customized command.

Whether the system permits modifying the solidified program and then solidifies is determined by the machine manufacturer.

4.6.4.11 Motor Drive Bit Parameter __ P405

【Motor drive】__bit parameter P405(password level：2)

d7	d6	d 5	d 4	d 3	d 2	d 1	d 0

The parameter sets the motor working state and whether there is X or Y axis control.

d7__Z motor direction;

d6__X motor direction;

d5__Y motor direction;

0： Positive.

1： Negative. // setting the motor direction parameter can change the motor rotation direction without changing others. The tool post actual movement direction is the same with the system defining direction.

d4__Z drive unit alarm level;

d3__X drive unit alarm level;

d2__Y drive unit alarm level;

0： LOW. // “Drive unit alarms” when Z, X, Y drive unit alarm input signal is LOW.

1：HIGH. // “Drive unit alarms” when Z, X, Y drive unit alarm input signal is HIGH.

d1__controllable axis Y

0： Without Y. // forbid using Y movement command.

1： With Y. // the controllable axis has Y, permits Y movement command.

d0__ controllable axis X

0： Without X. // forbid using X movement command.

1： With X. // the controllable axis has X, permits X movement command.

4.6.4.12 Parameters Related to Other Interfaces __ P412, P330～P332

【Relative interfaces】__bit parameter P412(password level：2)

d7	d6	d 5	d 4	d3	d2	d 1	d0

d7__external start/pause signal

0： None. // the external start and pause signals are invalid.

1： Have. // the external start and pause signals are valid.

d6__external feed hold knob

0： None. // the system external feed/spindle hold knob interface is invalid.

1： Have. // the system external feed/spindle hold knob interface is valid.

d5__pressure low check function

0： None.

1： Have. // the system has the pressure low check function; **P412_d4** sets the alarm level.

d4__pressure low alarm level

0： HIGH. // it is not connected with 0V.

1： LOW. // it is connected with 0V. when the system pressure low alarm check function is selected, the system has checked the pressure low alarm signal PRES is valid and the signal hold time exceeds P332 setting value, the system prompts "Pressure low alarm; at the moment, the axis feed pauses, the spindle stops and the automatic cycle cannot start.

d3__Call M72 when driver alarms.

0: Do not call.

1: Call. //The solidified M72 self-defined command is still be performed when driver alarms.

d2__ Perform M73 in MANUAL/AUTO resetting

0: Not performed. // In MANUAL/AUTO operation mode, the system only executes the standard reset function when controlling the reset key.

1: Performed. // In MANUAL/AUTO operation mode, when the reset key is performed, the system still executes the solidified M73 custom command after executing the standard reset key.

d1__ Perform M74 in MANUAL/AUTO ESP

0: Not Performed. // In MANUAL/AUTO operation mode, when the ESP alarm is valid, the system only executes the standard ESP function.

1: Performed. // In MANUAL/AUTO operation mode, when the ESP alarm is valid, the system still executes the solidified M74 custom command after executing the standard ESP alarm.

d0__miscellaneous function input pin macro programming

0： Do not permit. // forbid the statement programming to the defined standard input signal pin, execute the statement programming to the "UI" pin displayed in the diagnosis window.

1： Permit. // permit executing the statement programming to all input signal pin.

【Automatic lubricating start time(s) 】__auxiliary parameter P330：

The parameter defines the lubricating start time. (Unit: s)

【Automatic lubricating interval time(s) 】__auxiliary parameter P331

The parameter defines the automatic lubricating time interval (unit: s)

【Durable pressure low alarm time(s) 】__ auxiliary parameter P332

The parameter defines the hold pressure low alarm time. (Unit: s)

4.6.4.13 Auxiliary parameter __ P413～P418, P333 , P335, P027～P029

【Window language 】__bit parameter P413(password level：2)

d7	Reserved	Reserved	Reserved	Reserved	Reserved	Reserved	Reserved

d7__Chinese/English

0： Chinese. // display in Chinese.

1： English. // display in English.

[Communication interface] __Bit parameter P414 (Password level: 2)

d7	d6	reserved	reserved	reserved	reserved	reserved	reserved

d7__Baud rate selection (d6=0 enabled)

0： 9600.

1： 19200.

Bit selection	Baud rate
d7 =0, d6 =0	9600
d7 =1, d6 =0	19200
d7 =0. d6=1 d7 =1, d6=1	38400

d6__ Baud selection (Priority)

0： d7 enabled.

1： 38400.

[Display screen] __ Bit parameter P415 (Password level: 3)

d7	Reserved	Reserved	Reserved	d3	Reserved	Reserved	Reserved

d7__Parameter screen prompt

0: Close. //Close the range prompt of the parameter value;

1: Open //Open the range prompt of the parameter value.

d3__Diagnosis interface prompt

0: Close. //Close the prompt information in the diagnosis mode

1: Open. //Open the prompt information in the diagnosis mode.

[Function switch] __Bit parameter P416 (Password level: 3)

d7	d6	d 5	d 4	Reserved	Reserved	d 1	d0

d7__memorize current password level

0： do not memorize. // do not memorize the password level，the system displays Please

input user password when the system enters the parameter password input window.

1： Memorize. // **memorize the parameter** password level, enter the previous memorized parameter setting window.

d6__delete all program operation in EDIT working mode

0： Permit. // permit to delete all programs in EDIT working mode.

1： Forbid. // forbid to delete all programs in EDIT working mode.

d5__clear all offset operations in OFFSET working mode

0： Permit. // permit to delete all offset values in OFFSET working mode.

1： Forbid. // forbid to delete all offset values in OFFSET working mode.

d4__close drive unit enabling in JOG working mode

0：Forbid. // press “Delete” in JOG working mode, forbid closing the drive unit.

1：Permit. // press “Delete” in JOG working mode, permit closing the drive unit.

d1__Parameter display selection

0: All are displayed. //Both the alterable and unalterable parameters are displayed according to the parameter password level.

1: Alterable items are displayed. // Only the alterable parameters are displayed according to the parameter password level.

d0__modify machining programs in EDIT working mode

0： Permit. // program lock function is invalid, permit modifying the machining program on the operation panel in EDIT working mode.

1： Forbid. // program lock function is valid, forbid modifying the machining program on the operation panel in EDIT working mode, otherwise, the system alarms.

【Y axis reverse reset coordinates】__Reference coordinate parameter P027

【Z axis reverse reset coordinates】__Reference coordinate parameter P028

【X axis reverse reset coordinates】__Reference coordinate parameter P029

When Y axis is used for spindle or worktable reverse control, the reversed coordinate should be set within the range of this parameter. For example, when Y axis performs 360 degree reverse rotation, P027 should be set to 360.000; when Y axis moves, its tool nose coordinate and machine coordinate vary between 0～359.999.

When P027 is zero, this coordinate reset function is invalid; the range of setting is 0~9999.

【d value in G83】__auxiliary parameter **P335**

This parameter determines the d value in G83. The initial value is 2mm, refer to the G83 command of Section 4.21 in PROGRAMMING for details.

【Program line number automatic interval】__auxiliary parameter **P333**

This parameter determines the increment between two adjacent block numbers automatically generated in EDIT working mode, i.e. the difference between the two line numbers.

【Record alarm】__bit parameter P417（password level：2）

d7	d6	d 5	d 4	d3	reserved	reserved	reserved

d7__ Record parameter/tool offset alarm or not:

0： Record // Record the alarms in parameter/tool offset mode;

1：Not record // Does not record the alarms in parameter/tool offset mode;

d6, d5, d4, d3__ Record EDIT/program/ spindle/ tool change alarm or not:

0: Yes

1: No

This parameter P417 set whether the alarm is recorded. When it is set to 0, alarm is recorded; in following occasions alarm should be recorded: program changed, emergency stop, alarm in MANUAL/AUTO mode. Alarm messages can be divided into:

P417 bit parameter (record or not)	P418 bit parameter (display or not)	Message
d7	d7	Alarm in parameter/tool offset mode (E001~E099)
d6	d6	Alarm in EDIT mode (E101~E199)
d5	d5	Program related alarm (E201~E299, E601~E699)
d4	d4	M03/M04/M05 start/stop
d3	d3	Tool change
	d0	Emergency stop alarm, alarm in MANUAL/AUTO mode (E301~E499)

【Alarm display】__bit parameter P418（password level: 3）

d7	d6	d 5	d 4	d 3	reserved	reserved	d0

This parameter sets whether the recorded alarm message is to be displayed or not; when it is set to 0, alarm is displayed; messages for program change are all displayed; the displayed contents are:

d7__ display parameter/tool offset :

0: Yes // Displays all alarms in parameter/tool offset mode;

1: No // Does not display all alarms in parameter/tool offset mode;

d6, d5, d4, d3, d0__ Display EDIT/program/ spindle/ tool change/emergency stop in AUTO and JOG

0: Yes

1: No

【Note】

Alarm record is just a look-up function in special occasions; do not use it in normal circumstances.

4.6.4.14 Interface Parameter __ P500～P566

The system has the main devices including the spindle, the chuck, the tailstock, the tool post control and function, and the additional device control and check function. The former has the fixed signal pin, and the system does not set the fixed signal pin for the additional device determined by the manufacturer according to the actual condition of the machine.

When the unused signal of the main device has been released, the system adds the signal required by the additional device by setting the interface parameter, so, the system can control and check the additional device.

The system judges whether the interface parameter covers the input and output interface. P500～P510 are output interface parameter, and P511～P540, P557～P566 are input interface parameter.

Parameters P541～P556 are used for the multiple-station tool number signal encoding and for setting the tool number check signal.

The interface parameter value is not 0, i.e. covering the input or output interface; the setting value is the serial number of Common signal name, i.e. pin corresponding to the covered input or output Common signal name. The interface initial value is 0, i.e. the system does not use the parameter function, i.e. does not cover the pin.

When the input/output pin has been used by other functions, the interface parameter cannot be set as the occupied Common signal name and the system prompts: ［Parameter alarm］- Illegal I/O setting, input or output pin is occupied.

Search some pins have not been covered in DIAGNOSIS working mode, which are displayed in white, and whose names are displayed in the general signal names (UI** or UO**). Input/output pin serial number in the diagnosis window: increase from the number 1 from top to bottom, from left to right.

Standard definition and function of interface parameter are referred to **OPERATION, Appendix 4.6.5.8** Interface **Parameter List**.

【Example】

When P511 SAGT is set to 5, SAGT covers the input interface UI05; UI05 pin inputs the safety door check function.

When P506 M32O is set to 9, M32O covers the output interface UO09; UO09 pin outputs the lubricating control function.

4.6.4.15 Variable Initial Value __P600～P639

The initial value of variable parameter P600～P639 correspond to the variable initial value r001～r040; The concrete variable explanation is referred to **PROGRAMMING, Chapter 7 Statement Programming**.

4.6.4.16 **M87/M88** Related Parameter__**P342, P343**

Refer to the Section 4.4.3.5 *Spindle Orientation Control* of OPERATION for the detailed parameter explanation.

4.6.4.17 Self-defined Related Parameter __P344~P348

[Call M50~M72 before machining] __Miscellaneous parameter P344

Firstly, solidify the 254% program to call the self-defined command. Secondly, input the desired call self-defined command in the parameter P344, and then call this command before the program machining is performed.

[Call M50~M72 after machining] __Miscellaneous parameter P345

Firstly, solidify the 254% program to call the self-defined command. Secondly, input the desired call self-defined command in the parameter P345, and then call this command after the program machining is performed.

[Enter the Auto call M50~M72] __Miscellaneous parameter P346

Firstly, solidify the 254% program to call the self-defined command. Secondly, input the desired call self-defined command in the parameter P346, and then call this command when the system enters Auto working method.

[Retreat from the Auto call M50~M72] __Miscellaneous parameter P347

Firstly, solidify the 254% program to call the self-defined command. Secondly, input the desired call self-defined command in the parameter P347, and then call this command when the system is retreated from the Auto working mode.

[Call M50~M72 by manual 0 key] __Miscellaneous parameter P348

Firstly, solidify the 254% program to call the self-defined command. Secondly, input the desired call self-defined command in the parameter P348, and then directly call this command by “0” key in the manual working interface.

4.6.5 Appendix: Parameter List

4.6.5.1 Reference Parameter List

Parameter No.	Level	Name	Unit	Initial value	Range	Operator backup
P000	3	Z program reference point	mm	200.000	-99999.999～ +99999.999	
P001	3	X program reference point	mm	200.000		
P002	3	Y program reference point	mm	100.000		
P003	3	Z 2nd program reference point	mm	210.000		
P004	3	X 2nd program reference point	mm	210.000		
P005	3	Y 2nd program reference point	mm	110.000		
P006	3	Z 3rd program reference point	mm	220.000		
P007	3	X 3rd program reference point	mm	220.000		
P008	3	Y 3rd program reference point	mm	120.000		
P009	3	Z tool positive(tool nose) software limit	mm	8000.000		

P010	3	Z tool negative software limit	mm	-8000.000		
P011	3	X tool positive software limit	mm	8000.000		
P012	3	X tool negative software limit	mm	-8000.000		
P013	3	Y tool positive software limit	mm	8000.000		
P014	3	Y tool negative software limit	mm	-8000.000		
P015	2	Z positive(mechanical) software limit	mm	8000.000		
P016	2	Z negative software limit	mm	-8000.000		
P017	2	X positive software limit	mm	8000.000		
P018	2	X negative software limit	mm	-8000.000		
P019	2	Y positive software limit	mm	8000.000		
P020	2	Y negative software limit	mm	-8000.000		
P021	1	Z zero coordinate	mm	300.000		
P022	1	X zero coordinate	mm	300.000		
P023	1	Y zero coordinate	mm	200.000		
P024	2	Z zero offset value	mm	0	0～10	
P025	2	X zero offset value	mm	0	0～10	
P026	2	Y zero offset value	mm	0	0～10	
P027	2	Y axis return zeroing coordinate		0	0～9999	
P028	2	Z axis return zeroing coordinate		0	0～9999	
P029	2	X axis return zeroing coordinate		0	0～9999	

4.6.5.2 Motion Parameter List

Parameter No.	Parameter level	Parameter name	Unit	Stepper initial value	Servo initial value	Range	User backup
P100	2	Z rapid traverse speed limit	mm/min	5000	8000	1～30000	
P101	2	X rapid traverse speed limit	mm/min	2500	4000	1～30000	
P102	2	Y rapid traverse speed limit	mm/min	5000	8000	1～30000	
P103	2	Z lowest initial speed	mm/min	150	600	1～10000	
P104	2	X lowest initial speed	mm/min	75	300	1～10000	
P105	2	Y lowest initial speed	mm/min	150	600	1～10000	
P106	2	Z rapid feed acceleration/ deceleration time(ms)	ms	400	150	1～8000	
P107	2	X rapid feed acceleration/ deceleration time(ms)	ms	400	150	1～8000	
P108	2	Y rapid feed acceleration/ deceleration time(ms)	ms	400	150	1～8000	
P109	1	Z zero return low speed	mm/min	100	100	0～2000	
P110	1	X zero return low speed	mm/min	100	100	0～2000	
P111	1	Y zero return low speed	mm/min	100	100	0～2000	
P112	2	Initial speed in cutting feed	mm/min	100	200	1～9999	
P113	2	Max. speed limit in cutting feed	mm/min	4000	6000	1～15000	
P114	2	Linear acceleration/ deceleration time in cutting feed(ms)	ms	500	200	1～8000	
P116	2	Thread cutting acceleration time (ms)	ms	400	150	1～8000	
P116	2	acceleration/ deceleration time in thread cutting(ms)	ms	400	200	1～8000	
P118	2	G99 initial lines		0	0	0～30000	
P119	2	Delay time when orientation is switched to cutting（ms）	ms	100	100	0～1000	
P123	2	The top speed limit of Z axis zero return	mm/min	5000	8000	1～30000	
P124	2	The top speed limit of X axis zero return	mm/min	2500	4000	1～30000	
P125	2	The top speed limit of Y axis zero return	mm/min	5000	8000	1～30000	

4.6.5.3 Transmission Parameter List

Parameter No.	Parameter level	Parameter name	Unit	Initial value	Range	User backup
P200	2	Z backlash	mm	0.000	0～10.000	
P201	2	X backlash	mm	0.000		
P202	2	Y backlash	mm	0.000		
P203	1	Z command pulse multiplication ratio		1	1～9999999	
P204	1	Z command pulse division coefficient		1		
P205	1	X command pulse multiplication ratio		1		
P206	1	X command pulse division coefficient		1		
P207	1	Y command pulse multiplication ratio		1		
P208	1	Y command pulse division coefficient		1		
P209	1	Spindle encoder lines		1200	100～5000	

4.6.5.4 Auxiliary Parameter List

Parameter No.	Parameter level	Parameter name	Unit	Initial value	Range	User backup
P300	2	Max. speed of spindle M41	r/min	3000	1～99999	
P301	2	Max. speed of spindle M42	r/min	3000		
P302	2	Max. speed of spindle M43	r/min	3000		
P303	2	Max. speed of spindle M44	r/min	3000		
P304	2	Lowest speed limit of constant surface speed	r/min	100		
P305	2	Max. speed limit of constant surface speed	r/min	8000		
P306	2	Thread smooth speed borderline	r/min	1	1～9999	
P307	2	Thread spindle wave alarm	r/min	300	1～99999	
P308	2	Spindle JOG time 1(ms)	ms	0	1～99999	
P309	2	Speed in spindle JOG	r/min	200	0～99999	
P310	1	Covered line of spindle gear control		4	0～4	
P311	2	Frequency spindle gear shifting time 1 (ms)	ms	100	1～99999	
P312	2	Frequency spindle gear shifting time 2 (ms)	ms	100	1～99999	
P313	2	Spindle gear switch interval time(ms)	ms	100	1～99999	
P314	2	Output voltage in spindle gear shifting(mV)	mV	0	0～10000	
P315	2	Spindle stop brake delay time(ms)	ms	100	1～99999	
P316	2	Spindle brake output time(ms)	ms	1000	1～99999	
P317	2	Spindle max. speed limit	r/min	8000	1～99999	
P318	1	Tool post type(0__9)		0	0～9	
P319	1	Max. tool number		4	1～16	
P326	2	Spindle controlling pulse time(ms)	ms	10	1～99999	
P327	2	Chuck controlling pulse time(ms)	ms	10	1～99999	
P328	2	Tailstock controlling pulse time(ms)	ms	10	1～99999	
P329	2	M respond check time(ms)	ms	5000	1～99999	
P330	3	Automatic lubricating start time(s)	s	60	0～99999.999	
P331	3	Automatic lubricating interval time(s)	s	600	0～99999.999	
P332	3	Durable pressure low alarm time(s)	s	600	0.001～99999.999	

P333	3	Program line number automatic interval		10	0～100	
P335	3	d value in G83	mm	2	0～999.999	
P341	2	Cutting enabled when a percentage of rotation speed is reached		0	0～90	
P342	2	M87communication delay（ms）	ms	10	0～2000	
P343	2	M87communication address(decimal)		0	0～99999	
P344	2	Call M50~72 before machining		0	0~255	
P345	2	Call M50~72 after machining		0	0~255	
P346	2	Enter the Auto call M50~M72		0	0~255	
P347	2	Retreat from the Auto call M50~M72		0	0~255	
P348	2	Call M50~M72 by manual 0 key		0	0~255	
P350	2	Spindle clamping/releasing time (ms)	ms	0	0~9999	
P351	2	Servo OFF delay time (ms)	ms	0	0~9999	

4.6.5.5 Bit Parameter

Set the corresponding bit to 0 or 1 to realize the different control functions and to meet all kinds of requirements of different machines.

Parameter number	Parameter level	Parameter name	Initial value	Range	User backup
P400	3	Running setting	00000100	00000000 ～ 11111111	
P401	3	Efficiency setting	00000000		
P402	2	Safety setting 1	01000000		
P403	2	Safety setting 2	00100001		
P404	2	Debugging setting	00000000		
P405	2	Motor drive	00011100		
P406	1	Zero setting 1	00000000		
P407	1	Zero setting 2	00000000		
P408	2	reserve	00000000		
P409	1	Chuck tailstock (hydraulic system)	00000000		
P410	1	Spindle configuration	01000000		
P411	2	Precision compensation	00000010		
P412	2	Other interfaces	00000000		
P413	2	Interface language	00000000		
P414	2	Communication interface	00000000		
P415	3	Display window	10001000		
P416	3	Functional switch	00000000		
P417	2	Record alarm	00000000		
P418	3	Alarm display	00000000		
P419	2	Safety setting 3	00000000		
P420	3	Efficiency setting 2	00000000		

4.6.5.6 Variable Initial Value List

Parameter number	Parameter level	Parameter name	Initial value	range	User backup
P600～P639	3	r001～r040	0	-99999999～99999999	

4.6.5.7 Interface Parameter List

Parameter No.	Parameter level	Signal name	Function	Standard definition and function	I/O	Initial value	Range	Backup
P500	1	M21O	User command output		O	0	0～32	
P501	1	M23O	User command output		O	0		
P502	1	LMP3	Program run signal indicator 3	Applied to three-color light control(green)	O	0		
P503	1	LMP2	Alarm indicator control signal 2	Applied to three-color light control(yellow)	O	0		
P504	1	LMP1	Alarm indicator control signal 1	Applied to three-color light control(red))	O	0		
P505	1	MDLY	Machine electricity delay Power-on control signal		O	0		
P506	1	M32O	Lubricating control switch	Control lubricating switch on machine	O	0		
P511	1	SAGT	Safety door check	Check the machine safety door state	I	0		
P512	1	Dalm	Feed device alarm check	Check feed device state in M20	I	0		
P513	1	M41I	Gear shifting in-position signal	Use frequency spindle in-position check signal M41	I	0		
P514	1	M42I	Gear shifting in-position signal	Use frequency spindle in-position check signal M42	I	0		
P515	1	M43I	Gear shifting in-position signal	Use frequency spindle in-position check signal M43	I	0		
P516	1	M44I	Gear shifting in-position signal	Use frequency spindle in-position check signal M44	I	0		
P517	1	M91I	User command input		I	0		
P518	1	M93I	User command input		I	0		
P519	1	RM78	Tailstock forward in-position check	Use in using hydraulic tailstock	I	0		
P520	1	RM79	Tailstock backward in-position check	Use in using hydraulic tailstock	I	0		
P521	1	Wsp	External MPG emergency stop	Applied to external MPG	I	0		
P522	1	WsY	External MPG axis selection Y	Applied to external MPG	I	0		
P523	1	WsX	External MPG axis selection X	Applied to external MPG	I	0		
P524	1	WsZ	External MPG axis selection Z	Applied to external MPG	I	0		
P525	1	Wbk2	External MPG override 2	Applied to external MPG	I	0		
P526	1	Wbk1	External MPG override 1	Applied to external MPG	I	0		
P531	1	531c	Lubrication inspection alarm		I	0		
P532	1	G31I	G31 input check	Define G31 input interface，G31 function is valid	I	0		
P533	1	M61I	M61/M62 execution switch	M61/M62 foot switch	I	0	0~32 and 99	
P534		M63I	M63/M64 execution switch	M63/M64 foot switch	I	0		
P535		M65I	M65/M66 execution switch	M65/M66 foot switch	I	0		
P536	1	M67I	M67/M68 execution switch	M67/M68 foot switch	I	0		
P538	1	Z off	Z axis movement limit		I	0	0~32	

P539	1	X off	X axis movement limit		I	0		
P540	1	Y off	Y axis movement limit		I	0		
P541 ～ P556	1		No. 1 ～ 16 tool selection check signal	Used for multi-tool selection signal encode	I	0	0 ～ 255	
P557	1	M51I	M51/M52execution switch		I	0	0~32	
P558	1	M53I	M53/M54execution switch		I	0		
P559	1	M55I	M55/M56execution switch		I	0		
P560	1	M57I	M57/M58execution switch		I	0		
P561	1	Wfk1	External feedrate 1	It is used for the debugging of the cutting feedrate	I	0	0~32	
P562	1	Wfk2	External feedrate 2		I	0		
P563	1	Wfk3	External feedrate 3		I	0		
P564	1	Wfk4	External feedrate 4		I	0		
P565	1	Wf_H	External feedrate INH		I	0		
P566	1	Wf_P	External feedrate P		I	0		

【Note】

1) Bit parameter P409_d4 =1：when the tailstock control function is invalid; the tailstock forward in-position check RM78, RM79 interface are invalid.

2) Bit parameter P410_d6 =0：when spindle S gear shifting controls; gear shifting in-position signal M41I, M42I, M43I, M44I interfaces are invalid.

4.6.5.8 Parameter List Related to Command Forbidden

Some command names related to some parameter setting are forbidden; once the parameter setting makes the command name forbidden conditions are created, the system forbids all corresponding commands and functions. Parameters which are related to the command forbidden are as follows:

Command name	Command function	Note: commands are forbidden when the following are valid
M41～M44	Automatic spindle gear shifting control	P410_d6 =0：spindle S control: gear shifting
S05～S16	Spindle S control	P410_d6=0 and P410_d5=0： Only input S00～S04 in spindle gear shifting output
M47～M48		P410_d4=0：do not switch the spindle and Y
M32～M33	Lubricating function	Interface P506=0
M04	Spindle rotation CCW	P410_d2=1：do not use the spindle CCW signal output
M10～M11	Workpiece clamped/ released	P409_d7=1：chuck control function is invalid
M78～M79	Tailstock forward/ backward	P409_d4=1：tailstock control function is invalid
Commands related to Y		P405_d1=0：without Y
Commands related to X		P405_d0=0：without X
Pin programming command	r2001～r2032	P412_d1=0 ：forbid the pin programming in miscellaneous function input

4.6.5.9 Parameter List Related to Output Interface Release

The parameter setting can ensure the interfaces are not used for the standard signals, at the moment, the interface release can be used for the general output interface; the interface parameters can be defined to the released output interface. Parameters related to output interface release are as follows.

Standard signal name	Function	General I/O	Variable name	Note: the parameter defines the release to be the variable
M79	Tailstock backward	UO16	r2016	P409_d4=1: tailstock control function is invalid
M78	Tailstock forward	UO15	r2015	P409_d4=1: tailstock control function is invalid
M10	Chuck clamping	UO14	r2014	P409_d7=1: chuck control function is invalid
M11	Chuck releasing	UO13	r2013	P409_d7=1: chuck control function is invalid
TL+	Tool post CW output	UO12	r2012	P318=0: line-up tool P318=9：M60 customizes the command to tool change
TL-	Tool post CCW output	UO11	r2011	P318=0: line-up tool P318=9：M60 customizes the command to tool change
M8	Cooling ON	UO10	r2010	
M9	Cooling OFF	UO09	r2009	P410_d7=0: spindle control: level mode
MSP	Spindle brake signal	UO08	r2008	P410_d3=1: spindle brake signal output none
M3	Spindle rotation CW	UO07	r2007	
M4	Spindle rotation CCW	UO06	r2006	P410_d2=1: spindle CCW rotation signal needless
M5	Spindle stop	UO05	r2005	P410_d7=0: spindle control: level mode
S04/M44	Spindle gear output	UO04	r2004	P310<4: release the signal cover 0/1/2/3 channel
S03/M43	Spindle gear output	UO03	r2003	P310<3 release the signal cover 0/1/2 channel
S02/M42	Spindle gear output	UO02	r2002	P310<2: release the signal cover 0/1 channel
S01/M41	Spindle gear output	UO01	r2001	P310=0: release the signal cover 0 channel

4.6.5.10 Parameter List Related to Input Interface Release

The parameter setting can ensure the interfaces are not used for the standard signals, at the moment, the interface release can be used for the general input interface; the interface parameters can be defined to the released input interface. Parameters related to input interface release are as follows:

Standard signal name	Function	General I/O name	Variable name	Note: the parameter defines the release to be the variable
+LT	Z/X/Y positive limit	UI32	r1032	P404_d6=1: shield hardware limit alarm
-LT	Z/X/Y negative limit	UI31	r1031	P404_d6=1: shield hardware limit alarm
DecY	Y axis deceleration signal	UI30	r1030	P405_d1=0: without Y P406_d5=0: without Y machine deceleration switch
DecX	X axis deceleration signal	UI29	r1029	P405_d0=0: without X P406_d6=0: without X machine deceleration switch
DecZ	Z axis deceleration signal	UI28	r1028	P406_d7=0: without Z machine deceleration switch
SP	External pause signal	UI27	r1027	P412_d7=0: without external start/ pause signal
ST	External cycle start signal	UI26	r1026	P412_d7=0: without external start/ pause signal
MXZ1	Feed hold signal	UI16	r1016	P412_d6=0: without external feed hold knob
MXZ2	External spindle/feed hold signal	UI15	r1015	P412_d6=0: without external feed hold knob
RM10	Chuck clamping in-position check	UI14	r1014	P409_d7=1: chuck control function is invalid P409_d5=0: do not need the chuck respond check

RM11	Chuck releasing in-position check	UI13	r1013	P409_d7=1: chuck control function is invalid P409_d5=0: do not need the chuck respond check
TPS	Hydraulic tailstock foot switch input	UI12	r1012	P409_d4=1: tailstock control function is invalid P409_d0=1: no hydraulic tailstock foot switch input
SHL	Hydraulic chuck foot switch input	UI11	r1011	P409_d7=1: chuck function control function is invalid P409_d1=1: no hydraulic chuck foot switch input
PRES	Pressure low check	UI10	r1010	P412_d5=0: without pressure low check
TCP	Tool post locking in-position signal	UI09	r1009	P318=0: Line-up tool P318=9：M60 customizes the command tool change
T8	Tool post position signal T8	UI08	r1008	P318=0：Line-up tool
T7	Tool post position signal T7	UI07	r1007	P318=0：Line-up tool
T6	Tool post position signal T6	UI06	r1006	P318=0：Line-up tool
T5	Tool post position signal T5	UI05	r1005	P318=0：Line-up tool
T4	Tool post position signal T4	UI04	r1004	P318=0：Line-up tool
T3	Tool post position signal T3	UI03	r1003	P318=0：Line-up tool
T2	Tool post position signal T2	UI02	r1002	P318=0：Line-up tool
T1	Tool post position signal T1	UI01	r1001	P318=0：Line-up tool

4.7 OFFSET Working Mode

OFFSET working mode: the system executes the compensation to the tool offset. The system prompts the corresponding intellective prompt message for each operation. At the same time, the operator can press the prompt key hp1 at the right top, and the detailed offset operation key.

The relative settings, operation input formats and example descriptions are as follows: all required function keys are expressed with icons; all input letter keys, or digit keys are expressed with underline; the system prompting message is expressed with frame.

Press to cancel the mistaken input when the input letters or digits are wrong.

Press

to exit the current operation before confirmation when the operator executes some setting or some operation or man-machine dialog.

The system sets 64 groups of tool offset value T01～T64, each tool offset number corresponds to one group. Each group separately records Z offset value, X offset value, Y offset value. The operator can make the number of tool offset groups and the total number of used tools the same through manual tool setting. The other tool offset data can only be input from the keyboard. Whether the Y or X offset input is valid is set by bit parameter **P405_d1 and P405_d0 separately**; when **P405_d1=0**, and **P405_d0=0**， X and Y offset values cannot be operated, which is displayed in grey on the screen.

◆ Main functions in OFFSET working mode include:

☆ Select, modify, clear tool offset data;

☆ Transmit tool offset data between U disc and CNC system by USB interface;

Press

to enter OFFSET working mode as Fig. 4-7:

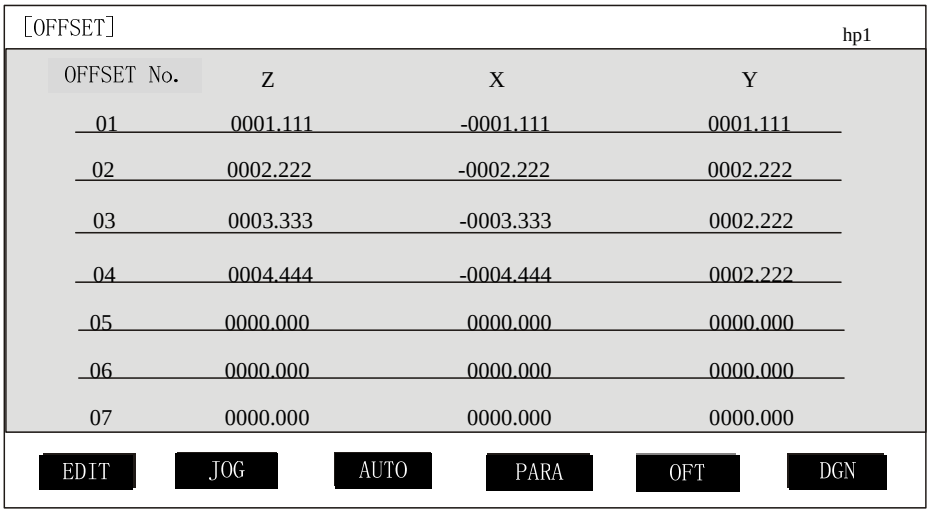


Fig. 4-7 OFFSET working mode

4.7.1 Tool Offset Value Search

The tool offset value search, i.e. search the required tool offset value as follows:

Method 1: scan

The operator can search the concrete content of each tool offset value in OFFSET working mode. Press , to search the previous or the next tool offset value. Press , to search the offset value on the previous page or the next page and there are 7 lines in each page.

Method 2: search

P + offset number + ENTER .

4.7.2 Input Tool Offset Data by Keyboard Key

X, Z, Y field range -8000.000 mm～8000.000mm;

Input tool offset data method by the keyboard: absolute input and relative input as follows:

Absolute input of tool offset data:

1) Select OFFSET working mode.

2) Press , , , to move the highlight to the required tool offset number; or use the search method to find out the required tool offset number; press to move the highlight to the required Z offset value, X offset value, Y offset value.

3) Press INPUT, or directly input the required data.

4) Input the required data by keyboard. Press to cancel the mistaken input and input the correct again.

5) Press ENTER to confirm the input, save the input data to the current selected too offset number parameter area.

6) The data which has been modified has "*" before it , which means the modification is successfully executed.

Relative input of tool offset data:

1) Select the OFFSET working mode; move the highlight to the required changing data.

2) Press INPUT or directly input the required data.

3) Input data by keyboard. Press to cancel the mistaken input data and input the correct. Press ALTER and the system operates the input data and the previous number value of the selected parameter. When the input is positive, the system adds the input data to the previous number value of the selected parameter to save the parameter area.

4.7.3 Tool Offset hp1 Function

Press hp1 in OFFSET working mode and the system displays the directory of the tool offset function prompts.

4.7.3.1 Communication of Tool Offset Data

Press hp2 key to transmit data through USB communication mode. The user can select the transmission direction as required.

1. **USB interface transmitting offset data:**

U disc root catalogue needs to create the file "C001OFT", the tool offset file name rule "OFT + file number (3-digit) + ".TXT" ten characters. The file number cannot be more than 254, otherwise, the system does not list it when it reads U disc. The tool offset file must be placed in C001OFT file folder. If there is no such a folder in the root directory of the U disc, the system prompts: No offset file in the directory after detection; after opening the tool offset directory page, the system can only send the tool offset files.

2. Standard format of TXT parameter file in U disc:

Refer to the format of tool offset files which are output to the U disc by the system.

4.7.3.2 Clearing Offset Values of Each Group in Offset Data

The steps of clearing the offset values of each group are as follows:

1）Select the Offset working mode.

2）Move the highlighted cursor block to the Y offset value to be cleared (move to X offset value if there is no Y axis, or move to Z offset value if there is no X axis)

3）Press DELETE key to clear the Z offset value, X offset value, or Y offset value of the tool offset number to be cleared.

4.7.3.3 Clearing Offset Data

P416_d5=0： the system permits to clear all offset value in OFFSET working mode, and the operators first presses hp1, and then Z to clear all offset data.

4.8 Diagnosis Working Mode

In DIAGNOSIS working mode: the system checks and displays the input/output interface state, spindle speed, encoder lines in real time. Press hp1 at the top right to learn about the diagnosis operation keys

◆ Main functions in DIAGNOSIS working mode include:

☆ Auto diagnosis function;

☆ display the external signal states of input/output(I/O)interface;

☆ Diagnosis of spindle control function input, output signal;

☆ Diagnosis of tool post control function input, output signal;

☆ Diagnosis of each axis hardware limit signal;

☆ Diagnosis of each machine zero(machine reference point)signal;

☆ Diagnosis of spindle speed and encode lines.

Press DIAGNOSIS to enter the DIAGNOSIS working mode and the display is as Fig.4-8:

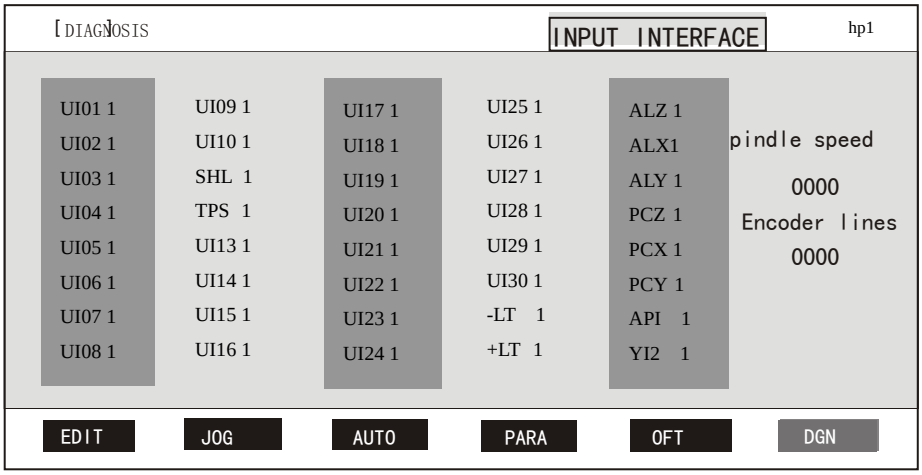


Fig. 4-8 Diagnosis working mode

4.8.1 Interface Signal Search

Firstly set **P415_d3**=1 to open the diagnosis page prompt, then you can view both the concrete content of each I/O in DIAGNOSIS working mode and the common signal name, hardware interface and pin number of each I/O in diagnosis working mode; when the parameter setting is closed, the operator cannot view the relative information.

Press , , , to search each I/O, and the searched I/O is highlighted. It can be displayed alternately in the input and output ports by or .

4.8.2 Interface Signal Name Display Explanations

The system has 23-channel switching input signals, 18-channel switching output signals; each signal has one name, which means to be the signal definition.

General signal name: the input signal names are UI01~UI32, the output signal names are UO01~ UO32, each signal corresponds to one pin. UI17~UI25 are not exported and UO19~ UO32 are not exported in interfaces.

Standard signal name: is called exclusive signal name. For a special machine, once some signal is covered by some special function, it has special name used for memory. The system set the standard exclusive signal name for the used signal of all functions.

In the diagnosis display window, the signals used by the special functions are expressed with the standard signal name; it means the set function is valid in the parameter. The initial parameter has used most miscellaneous functions, so the displayed most pin signal number is the standard signal name, the displayed unused signal are the general signal name.

Detailed explanation and connection method of general signal name and standard signal name are referred to **CONNECTION.**

4.8.3 Input Interface Diagnosis Explanation

In input interface diagnosis display, when one external signal is valid, the corresponding bit display is 0; when it is invalid, the display is 1. The input interface signal diagnosis is executed circularly; the system anytime displays the current signal state.

Input interface signal definition is defined by the parameter; changing corresponding setting of the parameter can change the input interface signal definition.

4.8.4 Output Interface Diagnosis Explanation

Each bit display is 0 in the output interface diagnosis; the corresponding bit output is valid. When the display is 1, the corresponding bit output is invalid. The output interface diagnosis display is the current each output big hold state. When the signal is pulse, the bit is still 1 although the output is valid.

Output interface signal definition is defined by the parameter, and changing the corresponding setting of the parameter can change the definition of output interface signal.

【Example】

Setting P511 SAGT to 1 means SAGT covers input interface UI01; and the system UI01 pin

inputs safety door check function.

In the first line of the leftmost column of the diagnosis input page is displayed the SAGT diagnosis information.

Setting P506 M32O to 18 means M32O covers the output interface UO18; and system UO18 pin outputs lubrication control function.

In the second line of the third column (from left) of the diagnosis input page is displayed the M32O diagnosis information

4.8.5 Output Interface Operation Function

Move the cursor by the direction key to the required writing output signal, press 1 or 0 to change the output signal value where the cursor is, when it is different with the current value, it is displayed in red; and it is the same with the current value, it is displayed in yellow.

【Note】

When the diagnosis output interface operation is used in the system debugging, the parameter password level is more than 2 to perform the operations.

4.8.6 Spindle Encoder and Spindle Speed Check

The system can check and display the pulse per rev of the spindle encoder, and the check result automatically displays the encoder lines.

The encoder lines mean the pulse per rev of the encoder.

The spindle speed means the current spindle actual speed(unit: r/min).

【Explanations】

1) When the spindle is not started, the displayed spindle speed is zero.

2) On initial page in DIAGNOSIS mode, when the spindle is started, if the checked encoder lines are not consistent with the P209 spindle encoder lines, the system prompts: [Diagnosis check prompts]: the encoder lines are not consistent with the parameter.

3) When the spindle encoder rotates synchronously with the spindle, i.e. when the spindle rotates one circle, the encoder rotates one circle; otherwise, the checked spindle speed is not consistent with the actual value.

4) Check the diagnosis encoder PA, PB and the counter signal by “R” key. This function is used for the miscellaneous diagnosis when the encoder malfunction occurs. If the PA and PAB signals 0/1 are displayed alternately when the spindle rotates, and the counter counts circularly, which means the PA and PB signal are normal.

4.8.7 Diagnosis hp2 Function

Press hp2 in DIAGNOSIS screen, the following prompt is displayed:

```
Key 0 – See alarm message Key 1 –See color code
Key 2 –Go to keyboard test Key 3 –See CPLD pulse number
Key 4 – See version info.Key 5 –Memory import/export
Key ESC -–Exit
```

Key 0 – To see alarm message: display alarm record;

Key 1 – To see color code: Display 256 colors and codes; for example, 00 is black, FF is white;

Key 2 – Go to keyboard test: to test the keyboard;

Key 3 –See CPLD pulse number: Display X/Z/Y pulse number;

Key 4 – See version information: display system version information including software, CPLD version, and software version loading;

Key 5 – Memory import/export: The system memory is transmitted through U disc.

1. USB interface transmits memory data:

It is required to create a folder with name “C001RAM” in the root directory of U disc. The naming rule of the system RAM file includes 10 characters, which are “RAM”+File number（3 digits）+“.TXT”. The file number cannot be greater than 999, or the system does not list the file when reading the U disc. The memory file must be placed in the folder C001RAM. If there is no such a folder, the system prompts: No RAM file in the directory after detection. After opening the RAM directory page, the system can only send the RAM files.

【Note】

1）Key hp2 is not advised to use.

2）For safety, the memory and version updating can only be done when the password level is above 2.

4.8.7.1 Alarm Record Display

In DIAGNOSIS mode, press key hp2, then, press 0, the alarm record is displayed; shown in fig. 4-9;

Alarm record function: Record alarm information. Every operation corresponds to a prompt. Meanwhile, you can press key hp1 to see details.

4031 alarm messages can be recorded at most; when the number exceeds 4031, the new alarm will replace an old one. So you need to download the alarm record in case of that.

◆ Main functions of alarm record are:

☆ Search for times of power on, alarm message;

☆ Display the total alarm number; display the alarm number to be listed; display the current alarm number;

☆ Transmit alarm record to U disc through USB interface;

Total number: 890	Displayed amount: 220	number0220	hp1
NO.125	90:59:02	E346 Tailstock retraction detection overtime alarm	
	1001:20:59	M05 Spindle stop	
	2001:05:59	%028 Change machining program	
NO.126	00:01:59	E367 Chuck unclamping in-position signal abnormal	
	01:11:20	E606 M91 used illegally：M91 pin is not defined;	
	01:21:59	E302 Z Drive unit alarm	
NO.129	00:40:20	E303 X drive unit alarm	
	120:40:20	E317 X tool nose software limit alarm in negative direction	

Fig. 4-9 Alarm record display

【Example】

A new alarm record: 120:40:20 E317 X tool nose software limit alarm in negative direction

The times of power-on is No. 129 displayed in the last alarm record. It means that this alarm record (number 220) happens after 120 hours 40 minutes 20 seconds later after the 129th power-on. The alarm contents are: "E317 X tool nose software limit alarm in negative direction".

【Note】

1) **During program changing, the number of the program is displayed on the alarm record; for example, %028 in Fig. 4-9 means that after 2001 hours 05 minutes and 59 seconds of the 125th power-on, the changed program number is %028.**
2) **The maximum displayed duration is 99999 H 59 Min 59 Sec; the largest displayed times of power on is 999.**
3) **The current alarm record is highlighted and the scroll bar on the right side of the screen shows the current position among all the alarm records.**
4) **Alarm record is a special search function; do not use it in normal circumstances.**

4.8.7.2 Alarm Record Search

The alarm record search is to find a specified alarm record, to view some identical alarms and to count the total number of record that is equal to the current alarm number. There are three ways to find specified alarm record:

Method 1: Scan

View the record details in alarm record screen. Press or to view last or next record. Press or to view the last page or next page. There are 8 lines in one page at most.

Method 2: Positioning

P+ **record number** + **Enter** (record number is the sequence number of alarm record).

Method 3: others

F + **search type** + **number** + Enter

The search type includes: E—through error number, M—through spindle command, T—through too change number, P—through program number; Press E, M, T, P respectively to enter different search types.

You can also view the identical alarm records by pressing or .

Press to count the quantity of records that are identical with the current alarm records; the display window will disappear immediately if other keys are controlled within 5 seconds.

[Related parameter]

The related parameter with the alarm record: P417 and P418.

4.8.7.3 Alarm Record hp2 Function

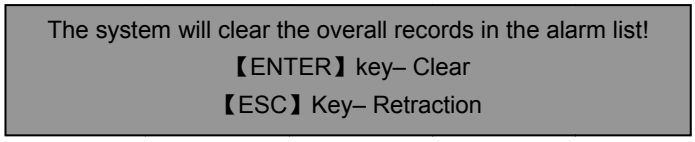
By pressing hp2 in alarm record screen, the data is transmitted by USB communication mode. The user needs to select the transmission direction.

1. Through USB interface

The alarm record that are transmitted through USB interface are stored in folder “C001ERR” in the root directory of U disc. The file should be named according to the rule “ERR + file name (3 characters) + TXT” 10 characters in total, and the file number should not be larger than 999.

4.8.7.4 The Deletion of Alarm Record

Press the DELETE key in the alarm record display interface, as follows:



The system clears the alarm records by the ENTER key in this page, and then the overall alarm records are cleared after performing, and return to the alarm record page; simultaneously, the “Without alarm record!” prompts at the left corner of this page. The alarm record will not be cleared press the Esc key, and then return to the alarm record page.

4.8.8 Machine Miscellaneous Function Control

Operate the miscellaneous function keys on the operation panel instead of the input commands in DIAGNOSIS working mode to execute the machine miscellaneous function.

Press

: The spindle rotates clockwise. **P410_d7=0**: LED indicator lights, M3 corresponding

bit in the output interface is displayed to 0. (**P410_d7=1**:LED indicator lights, M3 output in the output interface is first value, and after the pulse output is completed, M3 corresponding bit is displayed to 1).

Press

: The spindle stops. (**P410_d7=1**: the diagnosis output interface window displays M5.)

Press

: The coolant between ON/OFF shifts once. Bit parameter **P410_d7=0**, when the coolant ON, LED indicator ON, and the M8 corresponding position in the output port displays 0; when the coolant OFF, LED indicator OFF, and the M8 corresponding position in the output port displays 1. (In **P410_d7=1**, the coolant ON, LED indicator ON, and firstly output M8 in the output port is enabled; the M8 corresponding position display 1 till the pulse is output. When the coolant is OFF, LED indicator OFF, and firstly output M9 in the output port is enabled; the M9 corresponding position displays 1 till the pulse is output.)

CHAPTER FIVE USB SYSTEM COMMUNICATION

This system can transmit the part programs, system parameters, system software and tool offsets through USB interface; for the operation of the part program transmission, see Section 4.3.3 “Edit Operation Mode” in Part Ⅰ Operation; for the operation of the transmission of system parameters and system software, see Section 4.6.3 “Parameter Operation Mode” in Part Ⅰ; for the operation of tool offset transmission, see Section 4.7.4 “Tool Offset Operation Mode” in Part Ⅰ. For detail operation of receiving alarm record, see Section 4.8.7.3 Diagnosis Operation Mode.

5.1 USB Communication

The system supports USB communication mode, exchange the data between CNC and U disc by USB interface.

5.1.1 USB Operation

For the USB operation, the user directly inserts the U disc into the USB interface of the system panel, and the system automatically identifies and opens the U disc when the U disc requires creating the file and the file name in the root catalog according to the system file catalog. At the same time, the system window displays USB icon.

【Note】

1) After the U disc is used, the user must firstly press ESC to close the U disc before pulling out it, at the moment, the system displaying USB icon disappears, and then the user can pull out the U disc from USB interface, otherwise, the mistaken operation damages the system hardware and the U disc.

2) Must not pull out the U disc when its indicator flashes, because it is reading or writing the data, otherwise it can cause hardware damage or data loss.

3) The U disc has the write protect switch, and the user switches it before inserting the U disc and cannot do it when the U disc is working.

4) The user should reduce the storage content in the U disc; otherwise, it influences the communication speed between the system and the U disc. It is suggested the user not use large removable storage devices (exceeds 8G) on other USB interface; otherwise, it will cause damage to system hardware or removable storage devices.

5) When the U disc operation is fail, the user should press RESET and insert the U disc to operate it again.

5.1.2 USB File Catalog Requirements

In USB communication, for transmitting the different data, the system requires the USB file names are different, and the user creates the file name in the U disc catalog as follows:

Data / Name category	Part program	Parameter data	Offset data	System software upgrade
File name in U disc root catalog	C001PRO	C001PAR	C001OFT	C001DATA
File name in folder	CNCxxx.TXT	PARxxx.TXT	OFTxxx.TXT	DATAxxx.TXT
xxx range in file name	0~254	0~999	0~999	0~254

Data / Name category	Memory whole update	Alarm record	Syst0em RAM
File name in U disc root catalog	C001MEMO	C001ERR	C001RAM
File name in folder	MEMOxxx.TXT	ERR xxx.TXT	RAMxxx.TXT
xxx range in file name	0～254	0～999	0～999

PROGRAMMING

CHAPTER ONE PROGRAMMING FUNDAMENTAL

The automatic machining of CNC machine is the course of edited part programs automatically running. The programming is defined that the drawing and the technology of machining workpiece are described with CNC language and are edited to the part programs.

Here describes the definition of command and the programming mode of CNC part programs. Please read carefully these contents before programming.

1.1 Coordinate Axis and its Direction

The controllable axes of the system are defined as Z, X and Y axis, which realizes any two-axis or three-axis linear linkage; two coordinate axes perpendicular to each other form the plane rectangular coordinate systems **X—Z、X—Y and Z—Y;** as shown in fig. 1-1: the coordinate system in this manual defines X, Z axis according to fig. 1-1.

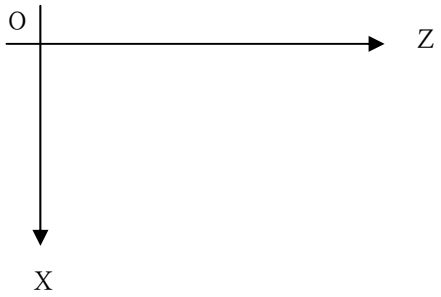


Fig. 1-1 X—Z rectangular coordinate system

X：It is defined to be perpendicular with the rotary centerline of spindle. X positive direction is the one that the tool leaves from the rotary center of spindle.

Z：It is defined to be coincident with the rotary centerline of spindle and Z positive direction is the one that the tool leaves from the headstock.

Y: Refer to X and Z; Y can switch with the servo spindle.

1.2 Machine Coordinate System, Machine Zero

Machine coordinate system is reference for the system to count coordinates, and is fixed on the machine.

Machine zero (or **machine reference point, or mechanical zero**) is a fixed point on the machine. Generally, it is set at the position of X, Z max. stroke with the machine zero check device, and do not use the machine zero return function or set bit parameters **P406_d7, P406_d6, P406_d5, P406_d4, P406_d3, P406_d2** to 0, i.e. “no machine zero” when the machine zero check device is not installed.

1.3 Program Reference Point

In machine coordinate system, the operator should set a position where the tool post stops, the tool changed is executed safely and installing workpiece is convenient. Program reference point is set when the tool post stops at the position which is called program reference point (or **program zero**). Program reference point coordinate is relative to machine coordinate system.

Once the reference point is defined, the tool can return to the reference point by executing the reference point return function in JOG working mode or AUTO working mode. Even if the system is switched off, the reference point still exists. If the stepper motor is employed, there is slight error caused by the motor vibrating after the system is switched on again. Execute the reference point return again to avoid the error. The program reference point is automatically set to X=200, Z=200 without setting the program reference point after the system is switched on for the first time.

1.4 Machine 2nd, 3rd Program Reference Point

In the machine coordinate system, the operator can also set the 2nd, 3rd program reference point. Their functions are similar to the program reference point, and the system has corresponding commands to move the worktable to the 2nd, 3rd program reference point.

1.5 Workpiece Coordinate System

The workpiece coordinate system is defined that some point on the workpiece is considered as the coordinate origin to create the coordinate system. Its axes are separately parallel with X, Z axis in the same direction. After the workpiece coordinates is created, all absolute coordinate values in programming are the position values in the workpiece coordinate system. Generally, Z workpiece coordinate system is set on the rotary center of workpiece.

According to the actual condition in programming, define the workpiece coordinate origin, i.e. the programming origin in the workpiece drawing and the coordinate origin of CNC system command. The workpiece coordinate system is created by setting a workpiece coordinate.

1.6 Positioning and Interpolation Function

The motion control mode that only controls the motion end point of one axis or multiple axes and the motion path during the movement is called the positioning control.

Interpolation is defined as the two-dimension (plane) contour formed by the resultant path of 2 axes moving at the same time, also called **Contour control**. The move axes controlled in interpolation are called the link axes, whose move amount, move direction and move speed are controlled simultaneously during the whole movement to form the desired resultant motion path.

The system is a two-axis linkage system whose link axes are X and Z, and it has the linear interpolation function.

1.7 Programming Coordinate

Programming coordinate is for workpiece coordinate system.

GSK96programming uses absolute coordinates (**X, Z, Y fields**), incremental (relative) coordinates (**U, W, V fields**) or compound coordinates(**X/W, U/Z, V/Y fields**) .

1.7.1 Absolute Coordinate Values

The absolute coordinate value is the distance to the coordinate origin, i.e. the coordinate position of the tool traversing to the end point as Fig. 1-2

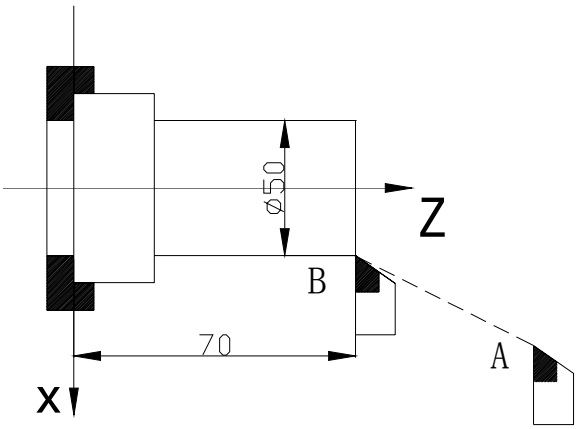


Fig.1-2 absolute coordinate value

Tool traversing from A to B uses B coordinate values as follows: **X 25 Z 70**. (Radius programming in X direction)

1.7.2 Relative (Incremental) Coordinate Values

The relative coordinate value is the distance from the previous position to the next one, i.e. the actual traversing distance of tool as Fig. 1-3:

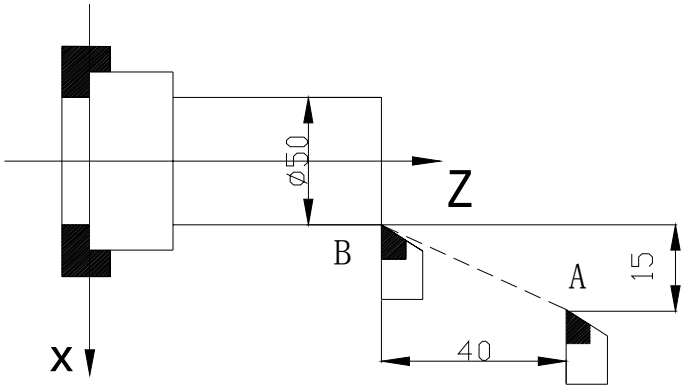


Fig.1-3incremental coordinate values

Tool traversing from A to B uses the incremental coordinates as follows: **U -15 W -40** (Radius programming in X direction)

1.7.3 Compound Coordinate Values

The incremental coordinates and the absolute coordinates can be applied at the same time, but one coordinate axis in one block can only be defined by one method, i.e. **X , W** or **U , Z** can be applied, but the **X,U** ,**Z,W** or **Y, V** cannot be applied.

Tool traversing from A to B as Fig. 1-3, X is applied with the absolute coordinate and Z with the incremental coordinate as : **X 25 W - 4 0**.

CHAPTER TWO PROGRAM STRUCTURE

CNC command set edited according to the requirement of machine moving is named as **program**. According to the sequence of command, the tool traverses along the straight line and the circular arc, or the spindle starts/stops, cooling is ON/OFF. The sequence of command is edited according to the technology requirement of workpiece.

2.1 Character

Character is the basic unit to compose the program. The character includes English letters, digits and other symbols.

English letters are address character of each command or data：**D E F G H I J K L M N P Q R S T U VW X Y Z r**

Digit is the specific data of each address character: **0，1，2，3，4，5，6，7，8，9**

Macro：**If then else and or = <> ()**

If then else : select statement; example：**If(x>y) then (z=0) else(z=1);**

and ：Logic and;

or：Logic or;

> ：More than;

< ：Less than;

= ：Equal to;

Sign：**% - + * . 凵 / ;**

%：Start sign of program name;

— ：Negative data or reduction;

+ ：Addition;

*****： Multiplication or modification;

. ： Decimal point;

/ ： Division in assignment statement, block skip sign in others 凵：blank space.

; ： Comment

2.2 Field

A field consists of an address character and the following numerical command. For example: N000 X12.8 W-23.45. Rules as follows:

- Each field must have an address character (English letter) and the following number character string.
- The invalid 0 of digital character string can be omitted.
- The leading zero of command can be omitted. For example: G00 can be written to G0.
- The positive sign must be omitted, but the negative sign must not be omitted.

2.3 Block Number

Block number is divided into two formats, i.e. it is with line number and without line number; block number is a line number of block number which is automatically created and also manually input and modified. A block number consists of the letter N and the following 4-digit integer (range: 0000-9999) . Block sequence can be at will, its interval also can be unequal. It is suggested to increase or decrease the block number progressively based on programming sequence in order to conveniently search or analyze programs.

When manual input is executed, block number N * * * * (* is 0～9) is directly input at the beginning of program line. When the integer following block number N is modified, please refer to the block content modification mode in OPERATION 4.1 *Edit Operation Mode* to modify the integer of block number.

When automatically creating a block number is executed, and **P333** is not set to 0, the line number can automatically create, otherwise, it cannot create automatically the line number. Please refer to **OPERATION 4.6 Parameter Operation Mode** about parameter setting; each program includes many blocks, and each block begins with block number" N**** ". After a new program is created, the system automatically creates the first block number " N0000 " , and after each block is input to press ENTER, the system automatically creates the next block number. The program number increment is defined by **P333** content. The system automatically creates block number for increment according to 1/4 integer of **P333** content.

【Note】

1) When the system finds that current row has no line number, it will automatically create "N1000".

2) When the commands M98, M97, M91, M92, M93, M94 related to block number are executed, there should be no repetitive block number, otherwise, the system will alarm. A block number can be repetitive without executing the above commands.

2.4 Block

A block with line number is composed of block number and many words; a block without line number has no block number. One block can contain 255 characters at most (including space between words). It is necessary to have the block number generated automatically by the system and can be modified in EDIT working mode.

A block can have multiple or no words. When a block has many words, one or more blank space must be input between words.

A complete block is as follows:

N0120 G1 X130 W-40 F50 Enter

N0120 block number

G1 ready function

X130 W-40 motion data

F50 motion speed

Enter end of block, displayed in the screen. But each program ends after pressing ENTER.

【Notes】

1) Each word of block is separated with a blank space generated automatically by the system, but it is necessary to input the blank space manually by user when this system cannot distinguish words.

2) A word can be placed on any position in a block.

2.5 Block Skip Symbol and Comment

Insert skip symbol "/" or comment symbol ";" before the block which needs not to be executed (and not be deleted) during execution. Then the system skips this block in the program being executed.

When a comment symbol is added behind each block, the user can add comments for each block; the comments are edited by only English letters and digitals on CNC, and can be edited by Chinese on PC. The CNC can display Chinese comments after programs are downloaded to CNC.

The system skips the block with "/"at the line beginning to execute the next one.

The system skips the block with ";"at the line beginning or block beginning to execute the next one.

Input ";"at the end of a line and then input simple comments.

【Notes】

1) Press W to insert ";" or "/".

2) The block contents behind ";" become green.

3) The first line of the block adds "/", and the contents of this block will become green.

4) The block in the cursor displays light green other than the 1st item that is regarded as prompt; the blocks in other items are orange.

5) The Chinese character notes followed with **";"** must be input by USB, the system keyboard can not input, but the system can display the Chinese characters.

6) Note that the inputted ";" is half-angle input format by USB, and the system does not support the full-angle input format.

2.6 Program Structure

A **block** consists of commands arraying of one or several technology operations in the course of machining. A part program consists of some blocks according to the machining technology orderly. A **block number** (**line number**) is used for identifying blocks. A **program name** is used for identifying programs.

Each part program consists of one program number and blocks. A program contains 9999 blocks at most. A block number is composed of N and the following 4-digit integer. A program name is comprised of % and program number (3-digit integer). General structure of program is as follows Fig.2-1.

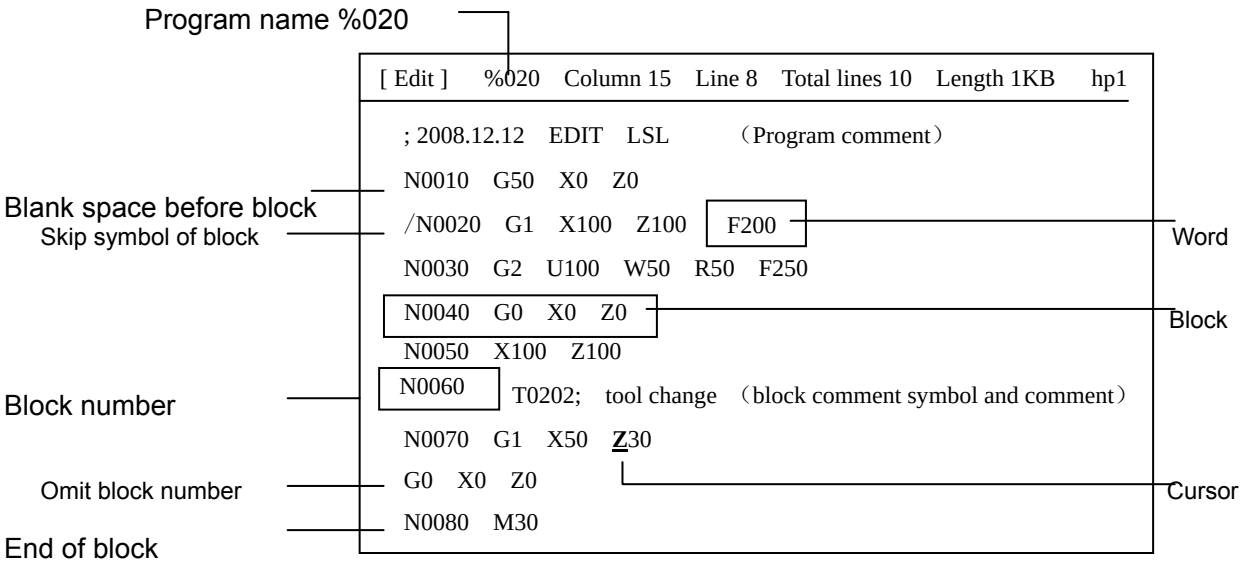

Program name %020
[Edit] %020 Column 15 Line 8 Total lines 10 Length 1KB hp1
; 2008.12.12 EDIT LSL （Program comment）
N0010 G50 X0 Z0
/N0020 G1 X100 Z100 F200
N0030 G2 U100 W50 R50 F250
N0040 G0 X0 Z0
N0050 X100 Z100
N0060 T0202; tool change （block comment symbol and comment）
N0070 G1 X50 Z30
G0 X0 Z0
N0080 M30
Blank space before block
Skip symbol of block
Block number
Omit block number
End of block
Word
Block
Cursor

Fig. 2-1 Program structure

A program consists of program comments, blocks; each block begins with block number (it can be omitted), follows characters, words; a block has block skip symbol and comments.

【Example】

Program(program name：%012)	Remark
N0000 G50 X200 Z300 G00 X200 Z200 N0020 T11 N0030 ; G00 X100 Z100 /N0040 G02 X150 Z150 R50 N0030 G01 X100 Z100 ; 2008 N0050 M30	Blocks containing N**** are the blocks with a line number, blocks not containing N**** are the blocks without a line number; Home of each line is a blank space; There is a blank space between line number and code for the program with line numbers; / Block skip ; The followings are comments.

CHAPTER THREE MSTF COMMANDS AND FUNCTIONS

This chapter introduces in details all **MSTF** functions and explanations of this system.

3.1 M — Miscellaneous Function (Command List)

M function is used to control some operations ON/OFF of machine and run sequence of machine program, and consists of address symbol M and its following two-digit integer. The used M functions in the system are as follows:

Command	Function	Programming format	Explanation
M00	Pause to wait for starting	M00	Press CYCLE START
M02	End of program	M02	Return to No.1 block
M20	End of program	M20 L	Return to No. 1 block to execute cycle machine, L being cycle machine times
M30	End of program for spindle OFF and cooling OFF	M30	
M03	Spindle CW	M03	
M04	Spindle CCW	M04	
M05	Spindle OFF	M05	
M08	Cooling ON	M08	
M09	Cooling OFF	M09	
M10	Workpiece clamp	M10	
M11	Workpiece release	M11	
M12	Chuck output signal cancel	M12	
M13	Output spindle clamping signal	M13	
M14	Retract spindle clamping signal	M14	
M32	Lubricating ON	M32	
M33	Lubricating OFF	M33	
M41	Spindle gearing No. 1	M41	
M42	Spindle gearing No. 2	M42	
M43	Spindle gearing No. 3	M43	
M44	Spindle gearing No. 4	M44	
M78	Tailstock forward	M78	
M79	Tailstock backward	M79	
M80	Tailstock output signal cancel	M80	
M95	Workpiece count	M95 K	
M96	Call cycle execution	M96 P Q L	
M97	Program transfer	M97 P	Transfer entry block number specified by P
M98	Subprogram call	M98 P L	
M99	Subprogram return	M99	M98/M99 specify call times
M21	Set No. 1 user output to be valid	M21 D	Output signal keeps the time specified by D, and the signal will be cancelled after the time arrival
M22	Set No. 1 user output to be invalid	M22 D	
M23	Set No. 2 user output to be valid	M23 D	
M24	Set No. 2 user output to be invalid	M24 D	
M91	No. 1 user input is invalid until input is invalid.	M91 P	Transfer entry block number specified by P
M92	Wait till No. 1 user input is valid	M92 P	Transfer entry block number specified

			by P
M93	Wait till No. 2 user input is valid	M93 P	Transfer entry block number specified by P
M94	Wait till No. 2 user input is valid	M94 P	Transfer entry block number specified by P
M47, M48	Set the permission or forbiddance state of Y	M47, M48	
M86	Send an information based upon the ModBus communication agreement	M86 P J Q H	
M87	Spindle orientation control	Refer to the *Spindle Orientation Control* in Section 4.4.3.5 of OPERATION for details.	
M88	Retract the spindle orientation control		
M81～M84	User input/output condition control	Refer to the M81, M82, M83, M84 command introduction in Section 3.2 of PROGRAMMING for details.	
M50~M74	Self-defined command	Refer to the *Self-defined Command Programming* in Chapter Eight of PROGRAMMING for details.	

【Note】

1) Leading zero of M can be omitted; Example: M00 can be written to M0, M0 and M00 are valid.

2) Bit parameter P403_d3, P403_d2: Set the output state of M function when the ESP alarm is enabled or during reset.

3) M commands are not in the same block with other commands as follows:

M00, M02, M20, M30, M95, M96, M97, M98, M99;

M21, M22, M23, M24, M91, M92, M93, M94;

M13, M14

M47, M48, **M86, M87, M88;**

M81, M82, M83,

M50～M74 **(It can be shared a same block with F, S).**

3.1.1 M00 — Pause

【Command format】

M00 ; program pause

【Explanation】

M00 pauses not to execute the program and the system prompts “DWELL” in flash and program continuously runs after CYCLE START is pressed.

Press ESC to exit from program running.

M00 functions are different from that of CYCLE PAUSE key. M00 is used to pause before some block which is specified in advance, and CYCLE PAUSE is used to pause at random.

3.1.2 M02 — End of Program

【Command format】

M02 ; end of program

【Explanation】

M02 indicates that program ends, and the system returns to the first block to wait.

3.1.3 M20 — End of Program Cycle Machine

【Command format】

M20 L ; return to the first block to execute the cycle machine, L is the cycle machine times, range: 1 ～ 9999

【Explanation】

L is the cycle machine times (when L is equal to 3, the actual number of machined workpiece is 4).

The system defaults the limitless cycle machine when L is omitted.

M20 indicates that the program ends and the system returns to the first block to execute repetitively, and is used when the system or machine is checked.

3.1.4 M30 — End of Program Spindle OFF Cooling OFF

【Command format】

M30 ; End of program, spindle OFF, cooling OFF

【Explanation】

M30 indicates the program ends, spindle is OFF and cooling is OFF and the system returns to the first block to wait.

3.1.5 M03, M04, M05 —Spindle Control

【Command format】

M03 ; spindle CW

M04 ; spindle CCW

M05 ; spindle stop

【Explanation】

The system is on the M05 initial state after the system is turned on. M03/M04 output is enabled and held the spindle on the OPEN state during the execution of the M03/04. Close the spindle when performing M05. Cancel the signal output of M03/M04 when the system is ESP.

See Section 4.4.3.1 *Spindle Start/Stop Control* in **OPERATION** for details.

3.1.6 M08, M09 — Cooling control

【Command format】

M08 ; Cooling ON

M09 ; Cooling OFF

【Explanation】

The system is on the M09 state when it is turned on. Perform the M08 and it is enabled, and then open the coolant; perform the M09, cancel the M08 output and close the coolant. Cancel the signal output of M08 when system is on the ESP state.

Refer to the *Coolant control* of Section 4.4.4 of **OPERATION.**

3.1.7 M10，M11, M12 — Clamping/Releasing Workpiece, Cancelling Chuck Output Signal

【Command format】

M10 ; clamping workpiece

M11 ; releasing workpiece

M12 ; cancelling chuck output signal;(only some special check device can use M12)

【Explanation】

M11 is valid and M10 output is invalid when the system is turned on.

The parameter can set M10/M11 to be enabled / disabled, to be checked or not, to be pulse or level control output, and to be inner or outer control mode. Interlock between chuck and spindle is decided by **P402_d5**: **P402_d5=**0, they are interlocked; **P402_d5=1,** their interlock is released. See OPERATION, 4.4.7 Hydraulic Chuck Control Function.

3.1.8 M13, M14 —Spindle Clamping Signal Output and Retraction

The control servo spindle is clamped by external equipment in the static state using the M13/M14 command output signal, and the interlocking is established between movement and clamping.

Only one block can be performed when programming. It is on the M14 state when the system is turned on. And it holds when the system is reset.

[Code format]

M13; Spindle clamping signal output

M14; Spindle clamping signal retraction

[Related parameter]

P509: Spindle clamping output signal SCLP, parameter range: (0~32)

P350: Spindle clamping/Releasing time (ms), parameter range: (0~9999)

P351: Servo OFF delay time (ms), parameter range: (0~9999)

◆ **The execution procedure of M13 command:**

The spindle clamping signal SCLP output is 0, delay the P350 time; the servo OFF signal YO3 (SVF) output is 0, delay the P351 time.

◆ **The execution procedure of M14 command:**

Retract the servo OFF signal YO3 (SVF), delay P351 time; retract the spindle clamping signal SCLP, delay P350 time.

The interlocking relationship between the spindle movement and spindle clamping:

There is the interlocking relationship between the spindle start and spindle clamping when P410_d4=0. M13 cannot be performed when the spindle is started, and the M14 then can be started the spindle.

There is the interlocking relationship among the spindle start, Y axis and spindle clamping when P410_d4=1. The M13 cannot be performed when spindle starts and Y axis moves, also, during the M47/87 is performed. And then the M14 state can be started the spindle or move the Y axis; otherwise, the system alarms “E394: M13 state prohibits the Y axis movement command”.

3.1.9 M32, M33 — Lubricating ON/OFF

【Command format】

M32 ; lubricating ON

M33 ; lubricating OFF

【Explanation 】

After the system is turned on, M33 is valid and M32 output is invalid. Parameter P330 sets whether the system automatically controls lubricating.

See OPERATION, 4.4.9.2 Cooling Control.

3.1.10 M41, M42, M44, M43 — Spindle Automatic Gear Shifting Control

【Command format 】

M41 ; Spindle (variable-frequency type) automatically shifts to No. 1 gear

M42 ; Spindle (variable-frequency type) automatically shifts to No. 2 gear

M43 ; Spindle (variable-frequency type) automatically shifts to No. 3 gear

M44 ; Spindle (variable-frequency type) automatically shifts to No. 4 gear

【Explanation】

M41, M42, M43, M44 can be used only when the machine employs the variable-frequency spindle. See Section 4.4.3.3 Spindle S Command—Rotation Speed Control in Part Ⅰ Operation for details.

3.1.11 M78, M79, M80 —The Output Signal of Tailstock Forward, Backward and Cancellation

【Command format】

M78 ; tailstock forward

M79 ; tailstock backward

M80 ; cancelling tailstock output signal(only some special tailstock device can use M80)

【Explanation】

Whether M78, M79 are valid, whether they require response detection, and whether they are pulse control output or level control output can be set by parameters. The tailstock is interlocked with the spindle (by setting **P402_d3**). See Section 4.4.8 Hydraulic Tailstock Control Function in Part Ⅰ Operation for details.

3.1.12 M95 — Workpiece Counting

[Format code]

M95K ; The resolution range of the K is 1 or -1.

[Explanations]

K=1; During the command performance, the workpiece counter may automatically add 1 calculation if the M95 K1 command is being met.

K=1; During the command performance, the workpiece counter may automatically subtract 1 calculation if the M95 K-1 command is being met.

[Example]

During the production, there are 5 workpieces are machined feeding once, and the counter adds 5.

```
N0000 G22 L5              ; Begin to define the cycle body, and cycle for 5 times
N0010 G50 Z100.5      ; Firstly, change the Z-axis cutter into 100.5 regardless how much before.
N0020 G0 W100             ; Z axis retracts to100
N0030 G0 X100             ; X axis retracts to 100
N0040 T11 M03 S600        ; Change T11
N0050 G0 X0 Z102          ; The No.1 tool is positioned nearby the barstock.
N0060 G1 Z100 F1000       ; Close the barstock at low speed, radial facing: 0.5mm
N0070 G1 X18              ; Retract to the external circle
N0080 G1 W-10             ;Cut the excircle
N0090 G1 X19              ;Retreat from the excircle
N0100 G0 X100 Z200        ;Retract to the tool-change
N0110 T22 S350            ;Change T22
N0120 G0 X25 Z95          ;Rapid traverse nearby the workpiece
N0130 G1 X10 F500         ;Groove
N0140 G0 X25              ;Rapidly retreat from the workpiece
N0150 G0 X100 Z200        ;Retract to the tool-change
```

```
N0160 T44              ;Change T44
N0170 G0 X25 Z90.5     ;Rapidly position nearby the workpiece
N0180 G1 X0 F500       ;Cut off
N0190 M95 K1           ;The workpiece counter adds 1 after machining
N0200 G80              ;The definition cycle body ends
N1000 G51              ;Recover the set workpiece coordinate system at the first time in the
                        manual mode.
N1010 M05              ;Spindle stops
N1020 G0 Z100.5        ;Retract to the initial position of the striker point
N0130 M95 K-1          ;Workpiece counter subtract 1
N0140 M02              ;End-of-program, return to the 1st block waiting, workpiece counter adds 1
```

When the program performs once means the M02, M20 and M30 command of the program end, and then the machining counting adds 1. And therefore, in this circle body, add one M95 K-1 command before the program end; in this case, the workpiece number can be display correctly.

3.1.13 M96 —Cycle Execution Call

【Command format】

M96 **P**＊＊＊＊ **Q**＊＊＊＊ **L** ＊＊＊＊ ; call cycle execution

【Field】

P — inlet block number for calling program. The leading zero of the block can be omitted.

Q — call last block number. The leading zero can be omitted.

L — call times. Omit L or call it one time when L is 1. L value is 1～9999 times.

【Explanation】

Program group called by M96 can contain M96, M98/M99, G22/G80 which can be embedded.

M96, M97 are embedded to avoid that the program returns to M96 to again run one time after M96 is executed.

【Example】

Method 1	**Method 2**
N0010 G00 X100 Z100	N0010 G00 X100 Z100
N0020 M96 P70 Q80 L3	N0020 M96 P40 Q50 L3
N0030 G01 W-5	N0030 M97 P0060
N0040 U5 W-5	N0040 G00 U5 W-5 F300
N0050 W-5	N0050 G01 U5 W-5
N0060 M30	N0060 G01 W-5
N0070 G00 U5 W-5 F300	N0070 U5 W-5
N0080 G01 U5 W-5	N0080 W-5
	N0090 M30

Method 1: After M96 specifies the call specified program three times, the cursor returns to N0030 and continuously runs till the program ends.

Method 2: After M96 specifies the call specified program three times, the cursor returns to N0030 and then the

system uses M97 to continuously runs till the program ends. The results of method 1 and method 2 are the same.

3.1.14 M97 — Program Transfer

【Command format】

M97 P ; program transfer

【Field】

P —Transfer to block number. The leading zero of the block number can be omitted.

【Explanation】

M97 commands the program to transfer from the block to the program specified by P. Block number specified by P should exist in the block, otherwise the program alarms "E215：line number miss"

The block number specified by P cannot be M97 block itself.

Generally, the death cycle must not be created in M97.

【Example】

```
N0030  G00  X100
N0040  M98  P0060
N0050  M97  P0090
N0060  G01  U2
N0070  W-5
N0080  M99
N0090  M02
```

Execute N0050 to directly do N0090 instead of N0060.

3.1.15 M98, M99 — Subprogram Call and Subprogram Return

【Command format】

M98 P ＊＊＊＊ **L** ＊＊

M99

【Field】

P — Block which is in subprogram. Leading zero of the block number can be omitted.

L — Call times of subprogram. Omit L or it is called one time when L is 1. L is 1～9999.

【Explanation】

Some fixed sequence which appears repetitively in the program is taken as a subprogram, so it can be called instead of being compiled when it needs again.

When M98 calls the subprogram and there is M99 in the execution of the subprogram, the subprogram call ends and the program returns to the main program to call the next block.

The subprogram generally follows M02/M30 of the main program, and the last block of the

subprogram must be the subprogram return command M99. When the subprogram does not follow the main program, it must command M97 to transfer the program.

【Notes】

1）M98 is used together with M99, and the subprogram call one time instead of L(L＞1) times is executed when M98 is used alone.

2）When the program has M99 without M98, the program does not end till M99.

3）M98/M99 is executed in the subprogram call, i.e. they can be embedded; the embedding can up to 18-layer.

3.1.15.1 M99 Increase L Address Character

The times of the program call can be altered by M99 L command. The resolution range of L is 0~9999. L=0 means that the call modification times is 0, and the subprogram call ends.

[Example]

```
N0000 G00 X100 Z50
N0010 M03 S01
N0020 G0 X66 Z15 F500
N0030 M98 P0070 L5     ; Call subprogram for 5 times
N0040 G0 X100 Z50
N0050 M05
N0060 M02              ; Main program ends
N0070 G0 U-4
N0080 G01 Z-10 F80
N0090 U20 Z-25
N0100 G0 U2 Z1
N0110 U-22
N0120 if(r1005=0) then(P0140)   ;If the No.5 input pin is LOW, and then turn to N0140 block.
N0130 M99              ;Subprogram return
N0140 M99 L3           ;The modification of the subprogram call is 3 times
```

[Notice]

During the program performance, M99 L cannot return at the same exit with the M98 command; otherwise, the repeated circular call may occur. As follows:

```
M98  P＊＊＊＊  L＊＊
……
M99  L＊＊
```

3.1.16 M21, M22, M23, M24 —User Output Control

【Command format】

M21	**D**	; output of No.1 user output signal is valid (output LOW)
M22	**D**	; output of No.1 user output signal is invalid (output OFF)

M23 D ; output of No.2 user output signal is valid (output LOW)
M24 D ; output of No.2 user output signal is invalid (output OFF)

【Field】

D — signal hold time.(unit: s 0～9999.999). When D is omitted, the output signal is being held.

【Explanations】

M21, M22, M23, M24 have not the concrete definition specified the system, and the user can specify according to the concrete requirements. They separately correspond to the output signals defined by the two users, and the corresponding commands can change the states of output signals.

User 1, user 2 output signals are defined in the interface parameter (**M21O, M23O**). See **CONNECTION** after the output interfaces are defined.

【Notes】

1) M21, M22, M23 and M24 cannot be in the same block with other commands. Each of them must be specified in a separate block.

2) M21～M24 can contain parameter D: when M21～M24 are executed with parameter D, the time specified by D is delayed after corresponding output is executed, and then the previous output is cancelled. If the command has no D, the output signal is being kept.

3.1.17 M91, M92, M93, M94 — User Input

【Command format】

M91 P ;
M92 P ;
M93 P ;
M94 P ;

【Field】

P—Block which skips to the target block. It does not skip when P is omitted. The leading zero of the block specified by P can be omitted.

【Explanation】

The definitions of input signals which correspond to M91, M92, M93, and M94 are not confirmed, and are defined by the user according to the requirements. They separately correspond to input signals defined by the two users. User 1, user 2 input signals are defined in the interface parameter (**M91I, M93I**). See Connection after the output interfaces are defined.

When P is omitted, the system orderly executes the next block when the state of the check input signal meets the command requirement, and always waits when it does not meet.

P≠0：The system skips to the block specified P when the input signal state meets the command requirement, otherwise orderly executes the next block.

P is omitted:

M91：Check the state of No. 1 user. When the state is valid (the input terminal is connected with 0V), wait till the input is invalid.

M92：Check the state of No. 1 user. When the state is invalid (the input terminal is broken with 0V), wait till the input is valid.

M93：Check the state of No. 2 user. When the state is valid (the input terminal is connected with 0V), wait till the input is invalid.

M94：Check the state of No. 2 user. When the state is invalid (the input terminal is broken with 0V), wait till the input is valid.

when P≠0：

M91：Check the state of No. 1 user. When the state is valid (input terminal connects with 0V), skip to the block specified by P, otherwise the next block is executed.

M92: check the state of No. 1 user, when the state is invalid (input terminal cuts off 0V), skip to the block specified by P, otherwise the next block is executed.

M93: check the state of No. 2 user, when the state is valid (input terminal connects with 0V), skip to the block specified by P, otherwise the next block is executed.

M94: check the state of No. 2 user, when the state is invalid (input terminal cuts off 0V), skip to the block specified by P, otherwise the next block is executed.

3.1.18 M47, M48 — Set Spindle Working State

【Command format】

M47 ;

M48 ;

【Explanation】

See Section 4.4.3.4 Setting Spindle Working State for details.

3.1.19 M86—Send an Information Based Upon the ModBus Communication Agreement

[Code format]

M86 P J Q H

[Field meaning and its range]

P: Equipment number：0-99999

J: Address：0-99999

Q：Allow using the r1~r199 variable in this filed, the resolution range of Q is (-9999999 ~ +9999999).

H：0000000-11111111

H_d0: 0—Write the data Q to equipment, 1—Read the content from the equipment address to the r4023.

[Explanations]

1) Do not input the execution in the manual mode;

2) When the equipment address content is read, the Q value is disabled. But the Q value cannot be omitted; otherwise, the system may alarm, and therefore, it is better to put any number within the data range.

[The execution procedure of M86 command]

Send information based upon the ModBus communication agreement, if the communication is failed, send it again.

The following prompt information may be generated during the execution of the command:

1) “Abnormal connection”: An abnormal response occurs after sending a signal; it may be either the error of the equipment number or address or the abnormal response from other interference.

2) “Fail to connection, without response”: Without response signal after sending signal; it may be the circuit malfunction or the incorrection of the baud rate.

3) “Retrying connection…”: The system is retrying to send

Note: 96 system does not lead out RS232 interface, it is necessary to use a special tool leading it. This operation should be performed by the qualified personnel.

[Example]

The following control program is compiled based upon a V&T frequency-converter controlling by GSK928TEa, and refer to this program when there are multi-frequency-converter; the alteration equipment number is corresponding to the system.

```
%254
N0000 -M51
N0010 M86 P1 J32768 Q10 H1   ;Read the data of the No.1 equipment 32768 address to the
                               r4023 (Q is disabled during reading data).
N0020 r20 = r4023            ; Save the read address content
N0030 r200 = 2000            ;The top speed S2000 of the motor
N0040 r201 = 100             ;The corresponding frequency 100HZ of the top speed
N0050 r203 = r200 / r201     ;The rotation numbers of each frequency
N0060 r204 = r151            ;The desired revolving speed (K) installed into the corresponding
                               variable
N0070 r205 = r204 / r203     ;Count the frequency to be output.
N0080 r205 = r205 / 1237     ;Frequency adjustment
N0090 r5 = r205 * 100000     ;Whether it is to be calculated according to frequency
N0100 r5 = abs(r5)
N0110 if(r204 < 0 ) then(P120) else(P150)   ;Perform downward, if the negative rotation speed is
                                              input; otherwise, skip to P150.
N0120 if(r20 = 0) then(P260)     ;If the original address is stop signal 0, and then deliver the
                                   reverse signal 1 and rotation speed.
N0130 if(r20 = 3) then(P240)     ;If the original address is positive signal 3, send a stop signal 0
                                   firstly, and then deliver a reverse signal 1 and rotation speed
                                   after delaying.
```

```
N0140 if(r20 = 1) then(P310)     ;Directly send a rotation speed to renew if the original address is
                                   reverse signal 1.
N0150 if(r204 = 0) then(P300)    ;Deliver a stop signal and 0 rotation speed if the speed input is
                                   0.
N0160 if(r204 > 0) then(P170)     ;It is downward performed if the positive speed inputs
N0170 if(r20 = 0) then(P280)      ;If the original address is stop signal 0, then send a positive
                                   signal and rotation speed.
N0180 if(r20 = 3) then(P310)      ;Directly deliver a rotation speed to renew if the original
                                   address is positive signal 3.
N0190 if(r20 = 1) then(P200)      ;Firstly send a stop signal if the original address is reserve
                                   signal 1,then deliver the positive signal 3 and rotation
                                   speed after delaying.
N0200 M86 P1 J32768 Q0 H0        ;As for the P equipment No. J address, the identical frequency
                                   should be consistent.
N0210 G04 D0.05
N0220 M86 P1 J32768 Q3 H0
N0230 M97 P310
N0240 M86 P1 J32768 Q0 H0
N0250 G04 D0.05
N0260 M86 P1 J32768 Q1 H0
N0270 M97 P310
N0280 M86 P1 J32768 Q3 H0
N0290 M97 P310
N0300 M86 P1 J32768 Q0 H0
N0310 M86 P1 J32769 Qr5 H0
N0340 M99
N0360 -M73                ;Stop the rotation by reset key
N0370 M86 P1 J32769 Q0 H0
N0380 M86 P1 J32768 Q0 H0
N0390 r151 = 0
N0400 M99
```

3.1.20 M87, M88 — Spindle Orientation Control

[Code format]

M87 ;

M88 ;

[Explanation]

Refer to the *Spindle Orientation Control* in Section 4.4.3.5 of **OPERATION**.

3.1.21 M50~M74 — Self-Defined Command

3.1.21.1 M50~M74 with Field K

[Code format]

M50 ；

M50 K ；(Input the execution in the manual mode)

[Explanations]

1) The K corresponds to **r150… r174 when** M50-M74 are performed; user can perform the branch judgment programming to the self-defined command by this function.

2) M50-M74 can be performed with field K, otherwise it is default K=0 (K= -9999999~+9999999).

[Example 1]

M51 K1300 ;No.1 unit head, rotation speed 1300

M52 K1000 No.2 unit head, rotation speed 1000.

Refer to the related examples for *M86 Command in Section 3.1.19 of* ***PROGRAMMING***

[Example 2]

The different states of 2-circuit signal output can be controlled by M51.

```
%254
N1000 -M51
N1010 if r151>0 then P1050
N1020 r2001=1
N1030 r2002=1
N1040 M99
N1050 if r151>1 then P1090
N1060 r2001=0
N1070 r2002=1
N1080 M99
N1090 if r151>2 then P1130
N1100 r2001=1
N1110 r2002=0
N1120 M99
N1130 r2001=0
N1140 r2002=0
N1150 M99
```

It is necessary to input the command in Manual or Auto program:

M51 K0 （or M51） ；Close 2-circuit signal output：r2001=1, r2002=1,
The K is set to 0 by default if it does not exist.

M51 K1 ；Output r2001=0，r2002=1

M51 K2 ；Output r2001=1，r2002=0

M51 K3 (It equals to K3 when K>3) ；Output r2001=0，r2002=0

3.1.21.2 In No.254 Program, When Compiling M50~M74 commands，with Parameter, but not Necessary.

[Code format]

-M50 I1.5.9…32 K1.3.6…32 ；

…

M99

[Field meaning]

I: Input port signal. Total 32 numbers from 1-32 followed I, wherein, the numbers should be separated by ".";

K: Output port signal. Total 32 number form 1-32 followed K, wherein, the numbers should be separated by ".".

[Explanation]

In the Manual or Auto working mode, the I/O signal observation interface appears by "Left" key after this command is executed. The function state of the input/output port signal followed with I/K can be observed in this interface. If there are several pages with the content in the display window, then view the I/O signal function state by the Up/Down page key.

3.2 M81, M82, M83, M84—User input/output condition control

M81, M82, M83and M84 are a group of condition control commands based on input/output signal state.

Representations of signal point, level and holding time are: for example: I8.1, Q17.0, R18.0, D5 etc.

1) Letters I, Q, R represents input signal, output signal and output signal state respectively (two states: when 0 is output, conducted circuit is formed externally; when 1 is output, it means high impedance state, conducted circuit cannot be formed externally;) D represents signal holding time.

2) The integral part after letters represents signal points, corresponding input signal UIxx or output signal UOxx pin; the range is 1~32 point.

3) **.0** behind the decimal point represents low level (0 can be omitted as it is default value); **.1** represents high level.

4) D: signal holding time; (unit: s range: 0~9999.999); when D is omitted, output signal remains the same.

5) Relevant parameter: bit parameter **P412_d0**. M81, M82, M83and M84are affected by this parameter.

3.2.1 M82—Output Control and Detection

【Code format. Example】

M82 Q17.0; UO17 pin outputs low level (conducted)

M82 Q17.1; UO17 pin output high level (high-impedance)

M82 Q17.0 D3; UO17 pin outputs low level, and output cancelled after delay time D.

M82 Q17.0 I8.0; UO17 output low level, and goes to nest till UI18 is at low level;

M82 Q17.0 I8.0 D3; UO17 pin outputs low level, after delay time D, cancels the output; goes to next command till UI08 is low level;

【Special explanation. Example】

M82 can be executed in following two formats:

Only two M82 commands can be executed in MANUAL working mode:

M82 Q17.0 D3 or M82 Q17.0

For example: M82 Q17.0 D5 ; UO17 pin outputs low level (conducted), then cancels the output after 5 seconds delay time. In MANUAL mode, it can be M Q 1 7 D 5.

3.2.2 M81—Control According to Input Signal State

【Code format. Example】

M81 I8.0 ; If the input UI08 pin is at low level, the system goes to the next command; otherwise, it keeps waiting.

M81 I8.1 ; If the input UI08 pin is at high level, the system goes to the next command; otherwise, it keeps waiting.

M81 I8.0 P1000 ; If the input UI08 pin is at low level, the system goes to P1000; otherwise, it goes to the next command.

M81 I8.0 Q17.0; If UI08=0, then, UO17 outputs 0, and system goes to the next command; otherwise, it keeps waiting.

M81 I8.0 Q17.0 D3; If UI08=0, then UO17 outputs 0, signal remains D3, and the system goes to next command; otherwise, it keeps waiting.

3.2.3 M83—Control According to Output Signal State

【Code format. Example】

M83 R8.0 ; If the input UO8 pin is at low level, the system goes to the next command; otherwise, it keeps waiting.

M83 R8.0 P1000; If the input UO8 pin is at low level, the system goes to P1000; otherwise, it goes to the next command.

M83 R8.0 Q17.0 ; If the No. UO8 pin is LOW of current state, the No.UO17 pin is output

LOW, and then perform the next one; otherwise, it keeps waiting

M83 R8.0 Q17.0 D3 ; If the No.UO8 pin is LOW of current state, the No.UO17 pin is output LOW, and signal keeps D3 and performs next one; otherwise, it keeps waiting.

3.2.4 M84 — Input Signal Inspection in the Specified Time

[Code format. Example]

M84 I8.0; If the input No.UI08 pin is LOW, then perform the next one; otherwise, keep waiting.

M84 I8.1; If the input No.UI08 pin is HIGH, then perform the next one; otherwise, keep waiting.

M84 I8.0 D3; Keep inspecting within D3, if UI08=0, then perform the next one; otherwise, it will keep detecting. If it does not inspect when the time is used up, the "E388：M84 inspection I8.0 signal overtime" alarms, and then the program is ended.

3.3 S function — Spindle Function

S and the following data are used to control the spindle speed and there are two modes as follows:

1) Spindle speed switch value control mode: Sx or Sxx, the system outputs the gear signal to the machine to realize the gear shift of spindle speed.
2) Spindle speed analog voltage control mode: Sxxxx specifies the actual speed, and outputs 0~10V analog voltage signal to spindle servo device or converter to realize the spindle speed change.

According to the machine's configuration with **P410_d6** setting, the spindle function is used to controlled multi-speed or variable-frequency motor.

3.3.1 Gear Shifting Controlling Spindle Motor

【Command format】

S01 ~ S04 ; 4-gear directly outputs, leading zero can be omitted.

S00 ~ S15 ; 16-gear BCD code outputs, leading zero can be omitted.

【Explanation】

P410_d6=0: S function controls multi-speed spindle motor; when it controls the multi-speed spindle motor, it selects the direct 4-gear control signal or outputs 16-gear BCD code signal. Parameter P410_d5 bit selects the output method of spindle multi-step control.

P410_d5=0: spindle control is the 4-gear direct controlling output S01～S04, each gear corresponds one output signal.

P410_d5=1 spindle control is 16-gear BCD code output S00～S15.

3.3.2 Spindle Controlling Variable-Frequency Motor

P410_d6=1: S function controls the conversion motor. The system outputs 0～10V DC analog voltage signal controlling inverter to realize the stepless speed regulating of the spindle motor.

◆ Gear control of **variable-frequency** spindle

【Command format】

M41 ; corresponding output signal S01

M42 ; corresponding output signal S02

M43 ; corresponding output signal S03

M44 ; corresponding output signal S04

◆Speed control of variable-frequency spindle

【Command format】

G96 ; Setting constant surface cutting state;

G96 S__ ; Setting constant surface cutting state, and specifying the surface speed value; range: 0～9999 m /min;

G97 ; Cancelling constant surface speed state;

G97 S__;Cancelling constant surface speed state, specifying speed value; range: 0～9999 r /min

S__ ; According to the current state; speed value or surface speed value cutting.

【Explanation】

1) In JOG and AUTO working mode, the system displays the actual spindle speed S_.

2) In JOG and AUTO working mode, the system displays the cutting constant surface control G96 S___m，or constant speed control G97 S___r.

3) Parameters are related to the spindle speed analog voltage control as follows:
Data parameter **P305:** max. spindle speed of constant surface speed;
Data parameter **P304**: lowest spindle speed of constant surface speed;
Data parameter **P300** ～ **P303:**max. speed of spindle in the spindle gear 1 ～ 4 (corresponding to M41～M44).

4) See Section 4.4.3.3 Spindle S Command—Rotation Speed Control in Part Ⅰ Operation for details.

3.4 T function — Tool Function

Machining one workpiece needs several different tools. The motor tool post with 4～16 tool selections can be controlled by the system. To avoid the error caused by the installation or the wear and tear, each tool is placed in the different position when it is cutting the workpiece; the tool change and the tool compensation are employed in the programming.

Tool setting operation is executed before machining receives the position offset data of each tool (referred to as **tool offset**). T command is executed in program running; the system automatically executes the tool offset. So, each tool in programming according to the workpiece drawing dimension is compiled instead of considering the position among each tool in the machine coordinate system. The deviation of machining dimension caused by tool wear can be changed according to the dimension deviation to modify the tool deviation.

【Command format】

Txx ____ the first 1- digit is the tool number, the second is the tool offset number;

Txxx ____ the first 1-digit is the tool number, the following 2-digit is the tool offset number;

Txxxx ____ the first 2-digit is tool number and the second 2-digit is tool offset number.

【Field】

Tool number range: Determined by **P319**(maximum tool number: 1～16); E.g. if **P319** is 4, the tool number is 0～4; if the input tool number is 0, it means keeping the current tool number.

Range of tool offset number: 0～64; if the input tool offset number is 0, it means cancelling the tool compensation.

【Explanation】

1) The system can select 16 tools and **P319** sets the most tool number.

2) When the system executes the manual tool setting operation, it automatically matches the tool offset number to the tool number and saves it to the tool compensation table. The system automatically identifies the relationship between tool setting record and tool number and tool offset number of the tool change command in the program; if inconsistency occurs, the system issues the Program Compound Check Alarm but does not lock the program execution.

3) The system executes the tool setting operation based on the trial cutting mode or fixed point mode in Jog operation mode. Refer to the Section 4.4.6 Manual Tool Setting in Part Ⅰ Operation.

4) When executing T command, firstly change the tool to the tool number specified by T command, then execute the offset value of T command. At this time, the system modifies the tool nose coordinates.

5) To enhance working efficiency, please specify T command and G0 command in the same block. E.g. G0 X100 T0202.

6) When T command is specified in the same block with G00/G01 command in AUTO working mode, the tool change process is firstly executed, then the tool is moved after adding the tool offset value to the coordinate of X (Z or Y) in G00/G01 command. The traverse speed is the rapid traverse speed in G00, and is cutting speed in G01.

7) Only when G0/G1 is in the same block with tool offset command, can the tool offset be added to the command value for execution.

8) Special rule: When the tool type is 9, T command automatically calls M60 command.

Therefore, T command must be specified in an independent block. It cannot be in the same block with other commands. Otherwise, an alarm occurs.

3.5 F function — Feedrate Function

It defines the feedrate of tool cutting i.e. the feedrate function (G98/G99, F command).

【Command format】

G98 **F**＊＊＊＊＊.＊＊＊ ; Feedrate per minute

G99 **F**＊＊＊＊＊.＊＊＊ ; Feedrate per rev

【Explanation】

Cutting feed: The system can control two axes (i.e. X, Z) to move simultaneously, making the tool motion path and the path defined by commands (straight line, arc) consistent, and the instantaneous speed in the tangent direction of motion path and F word consistent. This kind of motion control is called **cutting feed** or **interpolation**. The cutting feedrate is specified by word F. When the system executes the interpolation command (cutting feed), it divides the cutting feedrate specified by F into X, Z direction according to the programming path, and simultaneously controls X, Z instantaneous speed to make the vector resultant speed of speed in the twodirections equal to F command value.

$$f_x = \frac{d_x}{\sqrt{d_x^{\,2} + d_z^{\,2}}} \bullet F$$

$$f_z = \frac{d_z}{\sqrt{d_x^{\,2} + d_z^{\,2}}} \bullet F$$

F is the combined speed of vector of X/Y instantaneous speed;
d_x is the X instantaneous(d_t) increment,
f_x is the X instantaneous speed in X direction;
d_z is the Z instantaneous(d_t) increment ,
f_z is the Z instantaneous speed.

The command determines the cutting feedrate of the tool.

Feedrate function in feed per minute (G98) is expressed with F＊＊＊＊＊.＊＊＊.(range: 0～15000.000unit：mm/min)

Feedrate function in feed per rev (G99) is expressed with F＊＊＊＊＊.＊＊＊.(range: 0～15000.000unit：mm/r)

F is a modal value. Once specified, it is not necessary to input it again. It is the feed per minute (G98) state upon power-on, and the tool actual traverse rate is controlled by F and feedrate override.

Tool actual cutting speed = Fx feedrate override (feed per minute)

Tool actual cutting speed = Fx spindle speed x feedrate override (feed per revolution)

Reduction formula of feed between per rev and per min:**$F_m = F_r \times S$**

F_m: feed per min (mm/min);

F_r: feed per rev (mm/r);

S: spindle speed (r/min)

After the system is switched on, the feedrate is 0 and F value is reserved after F is commanded. The feedrate is 0 after F0 is executed. F value is reserved when the system resets and emergently

stops. The system supplies 16 steps for spindle override (0%～150%, increment of 10%). The feed override key on the system panel can regulate real-time, the actual feedrate override steps can be regulated in 0～150% and is reserved even if the system is switched off. The feedrate override regulation is referred to **OPERATION, 4.4.1.5 JOG Working Mode**.

【Relative parameters】

1) System parameter **P112**: the initial speed in cutting feed.

2) System parameter **P113**: max. speed limit of cutting feed.

3) System parameter **P114**: linear acceleration/deceleration of cutting feed.

4) System parameter **P118**: G99 initial lines.

【Note】

There is the uneven cutting feedrate when the spindle speed is lower than 1 r/min (in G99); there is the follow error in the actual cutting feedrate when there is the swing in the spindle speed. To gain the high machining quality, it is recommended that the selected spindle speed should be not lower than min. speed of the spindle servo or the converter.

CHAPTER FOURG COMMANDS AND FUNCTIONS

Here describes the functions and the explanations of all G commands of the system.

4.1 G00 —Rapid Positioning

The tool rapidly traverses to the specified position in G00.

【Command format】

G00 Z（W)_ X（U)_ Y（V)_	；3-axis rapid traverse
G00 Z（W)_ X（U)_	；2-axis rapid traverse (Z/X, Z/Y, X/Y)
G00 Z（W)_	；Z-axis rapid traverse (the single axis may Z, X, Y)

【Field and range】

X, Z, Y： absolute coordinates of end point; range: (-9999.999～9999.999) mm

U, W, V：relative movement from starting point to end point; range: (-9999.999～9999.999) mm

Use relative or absolute coordinates; the coordinates not to be moved can be omitted; G00 without coordinate fields cannot be specified.

【Explanation】

Z/X/Y can select any single-axis, two-axis or three-axis for the simultaneous movement.

If the tool change command and tool offset command are in the same block with G00, the tool offset value is added to G00 movement value, in order to improve the work efficiency. In general, the tool change and tool offset commands should be specified in the same block with G00.

G00 actual run speed is controlled by the rapid override. For example, when G00 rapidly traverses in the independent movement mode, the actual speed is set by **P100, P101, P102**.

Z (X, Y) actual rapid speed = P100 (P101, P102) × rapid override

The actual max. speed of the machine depends on its actual condition and matched motor. For details about parameters, please see the manual issued by the machine tool builder. G00 is a modal command, which can be omitted in the next block in which it is used. G00 can be abbreviated to G0, and G0 and G00 are equivalent.

【Relative parameters】

Parameters related to G00: **P100, P101, P102, P103, P104, P105， P106, P107, P108, P112, P114, P400_d3.**

Each axis can move at its respective rapid traverse rate or simultaneously rapidly move according to the proportion, and their movement mode is defined by **P400_d3**: **P400_d3**=0, each axis performs positioning at rapid traverse rate separately (independent movement mode) in G00; **P400_d3**=1, each axis performs positioning rapidly and simultaneously (interpolation movement mode) in the proportion mode in G00.

When P400_d3=0, the independent movement mode is executed as follows: (an example of Z axis)

① Speed raising stage: Z axis raises the speed at the initial speed of **P103**;

② Raise the speed to max. speed set by **P100**; the acceleration time in the speed raising stage is **P106**.

When each axis moves at the rapid speed, the actual speed displayed by the system is the resultant rapid traverse speed; when two axes or more move simultaneously, the actual rapid traverse speed displayed will be greater than the values set in **P100, P101**and **P102.**

P400_d3=1: interpolation movement mode is executed as follows:

① Raising speed stage: Raising the speed at the initial speed of **P112**;

② The acceleration time in the speed raising phase is **P114**. (It means the time the initial speed P112 takes to rise to 10000.000mm/min)

③ The resultant speed of the rapid interpolation is 10000.000mm/min. The movement axes rapidly perform positioning according to the proportion, and their actual speeds are limited by **P100, P101, and P102.**

【Note】

Ensure the tool is placed at a safe position to prevent the tools collision when G00 is executed.

【Example】

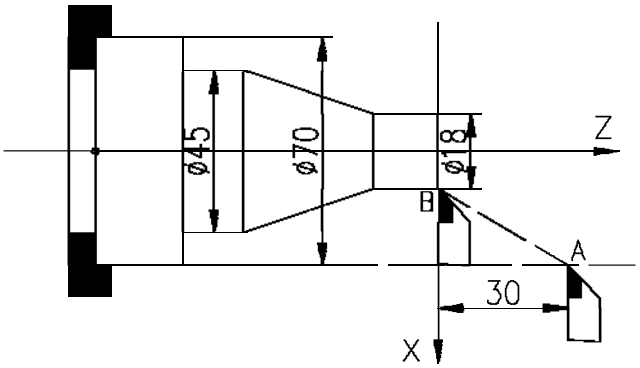


Fig. 4-1 G00 rapidly positioning

As Fig.4-1, the tool traverses from A to B.

Absolute programming: N0010 G00 X9 Z0 ;

Relative programming: N0100 G00 U-26 W-30 ;

4.2 G01 — Linear Interpolation

【Command format】

G01 Z（W)_ X（U)_ Y（V)_ F_；Z/X/Y 3-axis feed simultaneously

G01 Z(W)_ X(U)_ F_; Z/X two axes feed simultaneously. The two axes can be Z/X, Z/Y, X/Y do

G01 Z(W)_ F_; Z single-axis feed. The single axis can be Z, or, X or Y.

G01 command causes the tool to move along the line from current point to X（U）, Z（W）or Y（V）specified point at the set speed, and to reach the specified end point simultaneously.

【Field and range】

X, Z, Y: absolute coordinates of end point;

U, W, V: relative move amount from starting point to end point.

F: cutting feedrate. F value is modal, and can be omitted when it is not changed. At the same time, it is controlled by the feedrate override.

Range of X, Z, Y, U, W, V：(-9999.999～9999.999) mm

F range：0.001mm/min～15000mm/min

【Explanation】

Z/X/Y can execute single-axis, two-axis or three-axis feed simultaneously.

G01 traverse speed is specified by F and controlled by the feedrate override.

Actual feedrate = F × feedrate override

G01 is the modal command and can be omitted in the next block.G01 can be abbreviated to G1, and G1 and G01 are equivalent.

【Relative parameters】

Parameters related to G01: P112, P113, P114, P401_d5, P401_d4

Interpolation traverse execution as follows:

① Raising speed stage: raises speed at the initial speed of **P112**;

② The acceleration time of raising speed stage is **P114**; at the same time, the system checks whether the feedrate (Fx feedrate override) exceeds **P113** limit, if it does, the feedrate is **P113**.

P401_d5, P401_d4 set in the cutting machining: the system uses high-speed connection mode, continuous smooth transition or reducing speed to zero.

【Example】

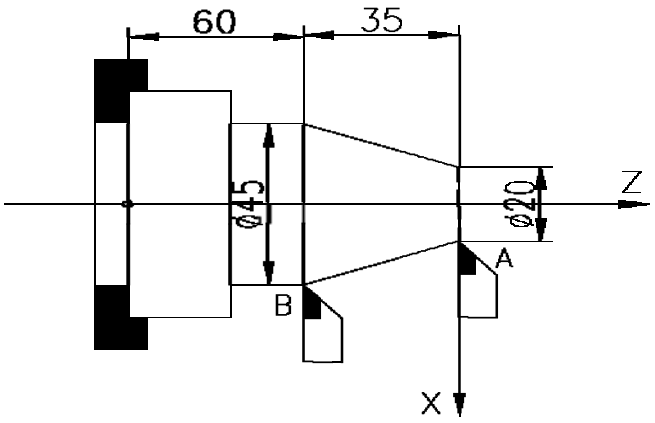


Fig. 4-2 G01 linear interpolation

The tool traverses from A to B in G01 at the speed 150 mm/min as Fig. 4-2:

Absolute programming: N0100 G01 X22.5 Z-35 F150;

Relative programming: N0100 G01 U12.5 W-35 F150 ;

【Note】

F cutting feed is executed with feed per minute(G98) except for special explanations; it is 0 when the system uses F programming, at the moment, the system alarms in running programs: feedrate is zero.

4.3 G06—Enter G06 Motion Mode with Single Axis；G07—Stop G06 Motion；G08—Enabling/Disabling G06Mode

G06/G07 command is usually used for controlling the long-time moving, indexing and stop of the rotation axis.

The definition of G06 command: The mode of enabling a coordinate axis to move at the specified F speed is called G06 motion mode in this manual. After entering G06 motion mode, the specified axis will move automatically till the specified position is reached or the stop condition is satisfied. During the execution of G06 motion mode, the user can perform any other operations to the system such as programming and program execution. Whether G06 is allowed for Z, X, Y is set by **P419_d3～P419_d1.**

【**G06 command format**】(The following example is described based on Z axis, and X and Y axes are the same as Z axis)

G06 Z（W)_ I8.0 F_ ； If there is a figure behind Z(W), it means the single-axis feed, and the axis stops after finishing the distance; if the low level of the input pin of UI08 is detected, the axis stops automatically.

G06 Z+ I8.0 F_ ； If there is a positive sign without figure behind Z, it means Z positive feed. The axis keeps moving until it reaches the soft/hard limit. If the low level of the input pin of UI08 is detected, the axis stops automatically.

G06 Z- I8.1 F_ ； If there is a negative sign without figure behind Z, it means Z negative feed. The axis keeps moving until it reaches the soft/hard limit. If the low level of the input pin of UI08 is detected, the axis stops automatically.

G06 Z F_ ； If there is neither a sign nor a figure behind Z, it means Z speed is changed to the set value (this format only refreshes the speed instead of moving the axis)

G06 Z（W)_ I8.0 F_ H_ D_ ； The number followed with Z(W) means single feed, it will be automatically stopped after executing; it will automatically stopped if the inspection of the No.UI08 input pin is LOW.

[Explanations]

I range: 0～32; “I” can be defaulted. Input the 0 followed decimal point with I means LOW (It may omit when it is set to 0, the system is set to 0 by default); 1 means High Level.

F range: (0.001～30000) mm/min; F can be omitted.

D_: This program can be completed after delaying, at the same time, the delay should be carried out; (Unit: S)

H _: It should be returned after performing, where it returns is determined by H; the H range: 00000000~11111111 (The system default value is 00000000 when it is omitted.)

H_d0=1: Return to this command start; (d0 priority)

H_d1=1: Return the G66 memory position; (If H_ d1=1, the G66 command should be in the block, otherwise, the system compilation can not be carried out.)

H_ d7=0: It stops immediately once inspect the I meets condition; H_d0=1, return this command start after stopping;

H_d7=1: It starts perform the G06 once inspect the I meets condition; if detect it again, the G06 command will not stopped.

[Example]

N0000 **G00 Z102 X120** ;

N0010 **G66** ; Memorize the current coordinate value Z102, X120

N0020 **G01 Z120 X130**;

N0030 **G06 Z150 I8.0 F300 H00000010 D3** ;The Z axis feeds to the Z150 based upon the F300 speed, and delay 3s after reaching, after that return to the Z102 memorized by G66.

【G06 Explanation】

1) Each G06 command can only start one axis feed; if another axis needs to be started, it is necessary to input another G06 command.

2) G06 stops feeding when the emergency stop button is pressed; When the RESET key is pressed, G06 motion mode is set by parameter **P419_d0.** When **P419_d0** is set to 1, the system executes the standard reset function by pressing reset key, which does not affect G06 feed.

3) When an axis enters G06 motion mode, the axis automatically stops when it meets the soft limit; if it meets the hard limit, whether it is decelerated to stop or stopped immediately is set by parameter **P402_d7.**

4) When G06 command is executed in JOG/AUTO working mode, the CYCLE DWELL key, external pause button are invalid for G06 feed.

5) Refer to G00 for the other relative parameters of G06 command.

6) The traverse speed of G6 is specified by F, which is not controlled by the feed override; and the max. speed of each axis is limited by parameter **P100～P102.**

7) The speed specified by G06 is valid only in this (or next G6) command, and it does not affect the other command speed in JOG or AUTO working mode; the other command speed does not affect the G6 speed memorized in the system either.

8) If the speed of G6 mode is not specified after system power-on, or F is omitted, the system presets the speed to F30.

9) When an axis is in the G6 motion mode, it is available to change the speed by using G6 command. E.g. G6 Z F500; which means the Z speed is changed to F500,

10) In the program, if Z axis is executing G6, and a new command relative to Z axis such as G6 or G1 is met, the system does not execute the next until the last G6 is executed. However, there are two exceptions: If G6 Z+ or G6 Z- is being executing, and a new G6 Z+ or G6 Z- is met, the system will proceeds to the next because they are completely the same. If a new command

only the G06 speed of which is different is met, they system immediately modifies the speed and then continues to execute the next command.

11) When G6 is executed, there is a small G6 command displayed in different color under the axis at the same time as axis movement, prompting the information about the axis movement.

【G07code format】

G07 Z ； Z single-axis automatically feeds, and the single axis can be Z, X or Y

G07 ； Without axis number, which means all axes automatically stop.

【G07 explanation】

1) The definition of G07 command: The specified axis automatically stops feeding, and the specified axis stops G6 motion mode.

2) G07 can only contain single axis or no axis number. It can be executed in JOG/AUTO working mode; if G07 is abbreviated to G7, it is equivalent to G7.

【Manual input of G06 and G07】

Take Z as an example: Once the parameter allows the axis to perform G6 motion, the user can input a complete G6/G7 command to control the axis in JOG working mode. For example, by inputting“G6 Z300 I8 F2 ENTER”, the Z starts to move.

The below common cases can be simplified by using the following input methods.

1) Key in: I Z ENTER ； Equivalent to G06 Z+

2) Key in: I Z- ENTER ； Equivalent to G06 Z-

3) Key in: I Z F300 ENTER ； Equivalent to G06 Z F300

4) Key in： I Z 0 ENTER ； Equivalent to G07 Z

5) Key in： I 0 ENTER ； Equivalent to G07

Note: The simplified input can only control the start and stop of the move axis and set the speed instead of controlling the positioning of the move axis.

【Keying in G06, G07 in AUTO mode】

In AUTO working mode, the user can employs the simplified input method to execute G6/G7 command before executing the program. During the program executing, the manual key-in control is valid if the axis has executed G6 before or is executing G6; and the manual key-in control is invalid if the axis has executed other commands other than G6 before. In some cases, you can also used G8 command to set whether it is valid or invalid.

【G08 command format】

G08 Z1 X0 Y1 ； Can be set to single-axis, two-axis or three-axis mode

【G08explanation】

The definition of G08 command: It means whether G06 can be executed manually after single block stop or pause in the following program. 1 behind the axis number means the axis enters G6 mode, and 0 means this axis forbids G6 mode.

E.g. G8 Z1 ； In the following program, Manual execution of G6/G7 is allowed for Z axis until it meets other motion commands.

G8 Z0 ； In the following program, manual execution of G6/G7 is not allowed for Z axis until the other G6 command is met.

【Integrated example】

In the example below, the Y axis is the servo rotation spindle; the program completes the outer circle turning, and then indexes to drill two holes, finally cuts off the workpiece.

```
N0000   G00 Z100 X100        ；Z/X return to the start point
N0010   G06 Y+ F1000         ；The spindle rotates at the speed of F1000
N0020   G00 Z90 X50          ；Z/X approach the workpiece
N0030   G01 Z50 F30          ；Turns the outer circle
N0040   G01 X51 F100         ；X retracts
N0050   G07 Y                ；Spindle stops
N0060   M21                  ；Unit head is in position
N0070   G00 Z75 Y10          ；Z moves to the first hole position, and the spindle rotates to 10
                               degree at the same time.
N0080   G01 X30 F80          ；X feeds to drill the first hole
N0090   G00 X51              ；X retracts
N0100   G00 Z60 Y130         ；Z moves to the second hole position, and the spindle rotates to
                               130 degree at the same time
N0110   G01 X30 F80          ；X feeds to drill the second hole
N0120   G00 X51              ；X retracts
N0130   M22                  ；Unit head retracts
N0140   G06 Y+ F2000         ；The spindle rotates at the speed of F2000
N0150   G00 Z55
N0160   G01 X0 F50           ；Cuts off
N0170   G07 Y                ；Spindle stops
N0180   M02                  ；End of program
```

4.4 G09 —(Y axis) The Shifting Command in Working State of Feeder

[Code format]

G09 Y0 or G09 Y1 ; The execution can be input in the Manual mode.

[Example]

G09 Y0 ; The Y axis can be moved in the Manual and programming; however the G6 cannot be performed in monitoring;

G09 Y1 ; The Y axis cannot be moved in the Manual and programming, but the G6 can be performed in monitoring.

The movement sequence of Y axis is compiled into monitoring program by G6 command if the Y

axis is used as the independent feeder control.

The feeder control can be divided into manual adjustment state and monitoring execution state.

The Y axis can be adjusted in the manual mode and programming after performing G09 Y0. At this moment, if the G6 occurs in monitoring programming, the monitoring will then refuse to perform and automatically close.

The execution state is monitored after performing G09 Y1; in this case, the Y axis cannot be adjusted in the manual mode.

G09 state is memorized after the system is turned off, avoid the malfunction of the feeder in manual mode.

4.5 Thread Cutting Command

The CNC system can execute many kinds of thread cutting function. The thread function includes: continuous thread cutting command G33, variable pitch thread cutting command G34, tapping cycle cutting command G32, the rigid tapping threading command G38.

The machine with the thread cutting function must be installed with the spindle encoder whose lines is set by P209. In thread cutting, the system moves the thread machining axes after it receives the one-turn signal of the spindle encoder, so the system can execute the roughing, finishing many times to complete the machining of the same thread without changing the spindle speed.

When the thread pitch is determined, the speed of the thread machining axes is determined by the spindle speed and is not related to feedrate override. When the spindle override control in the thread cutting is valid and the spindle speed changes, the pitch has an error because of the acceleration/deceleration. Therefore, never attempt to adjust the spindle speed or stop the spindle in thread cutting. Stopping the spindle may cause damage to the tool and workpiece. The thread cutting command can only be the single-axis (Z/X) programming.

4.5.1 G33 —Thread Cutting

【Command format】

G33 Z（W)_ P（E)_ Q_ H_ ；Z axis thread cutting

G33 X（U)_ P（E)_ Q_ H_ ；X axis thread cutting

G33 Y（V)_ P（E)_ Q_ H_ ；Y axis thread cutting

[Field]

P —— metric thread lead. P range：0.001mm～500.000mm.

E —— inch thread lead. E range：(0.060～25400.000) teeth/inch

X(U)/Z(W)/ Y(V) —— The Absolute/Relative coordinate of thread end;

X, Z, Y, U, W, V range：-9999.999mm～9999.999mm。

Q —— Initial angle, which is 0° if not specified; it is used to thread part. Q range：0°～360.000°.

H —— Command bit parameter (H_d7 ~ H_d2：reserved) ; H range：00000000～11111111.

H_d1: It selects the continuous thread machining raising/reducing speed mode and is 0 by default.

H_d1 = 0: In continuous thread cutting, between two neighboring threads, the long axis has a process of reducing the speed from machining speed to initial speed, and then raising the speed from initial speed to thread machining speed. The thread pitch length changes in the transition stage of speed raising and reducing

H_d1 = 1: In continuous thread cutting, between two neighboring threads, the machining speed of the current thread suddenly changes to the one of the next thread without the process of reducing the speed from machining speed to initial speed and then raising the speed from initial speed to thread machining speed. Therefore, this mode is not suitable if the pitch difference between the two threads is too big. Otherwise, the motor may step out.

【Relative parameters】

Parameters related to G33:**P100~P108, P113, P116, P209, P306, P307, P403_d0.**

Take Z as an axis thread for example, the relationship between G33 and relative parameters:

① Before the system enters raising speed stage, it firstly checks whether the thread cutting speed exceeds **P113**, if it does, the system issues an alarm to terminate the thread machining;

② Raising speed phrase: Z axis raises speed at **P103** initial speed;

③ Acceleration time of raising speed stage is **P116**;

④ The boundary of two thread machining modes is **P306**;

⑤ During thread cutting, the system detects the spindle speed through spindle encoder line number P209. If the spindle rotation speed exceeds P307, an alarm occurs after the current thread machining block is finished, and then the machining is suspended; the next step of machining procedure can be continued if press the CYCLE START key;

⑥ **P403_d0=0:** the system does not check whether the spindle speed is stable in the thread machining.

P403_d0=1: the system checks whether the spindle speed is stable in the thread machining.

【G33 execution process】

G33 execution process is as follows (taking Z axial thread cutting for example):

① Check the spindle speed, count the reference speed of thread machining according to the rotation speed and the lead P. When the reference speed exceeds P113 (max. cutting feedrate), the system issues an alarm. Before this, the user must ensure the spindle speed is stable; otherwise, there may be confused machining teeth.

② Check one-turn signal of coder from the initial angle; Z axis raises speed from the current position A (G33 starting point), which makes the traverse speed reach the reference speed, At the moment, Z moves to B; the lead of AB section maybe not standard (in the actual machining, the front of AB section must reserve an idle distance which does not touch the workpiece), the higher the reference speed is, the longer AB section is.

③ The system follows the spindle speed from B to execute the cutting to C; the cutting speed

changes with the spindle speed; the lead of BC section is standard.

④ Z reduces speed from C till D position Z(W); CD lead maybe not standard, CD length is equal to AB section (when the reference speed is lower than the initial speed, raising/reducing speed is not required, the lengths of CD and AB are 0).

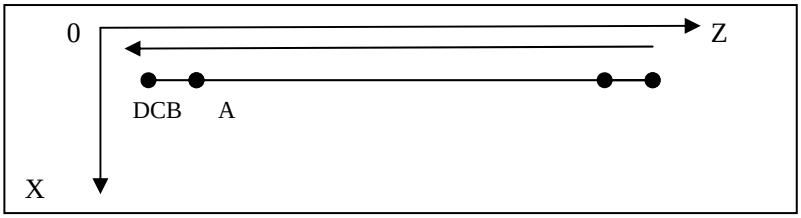


Fig. 4-3 G33 path

【Explanation】

1) G33 command can machine the inch system single-axis screw thread.

2) In the spindle rotating clockwise, the positive cutting is right-hand thread, and the negative is the left-hand thread, and the spindle rotates counterclockwise, they are reverse.

3) Generally, the machining repeats the thread cutting many times in the same path from the roughing to the finish turning when the thread is machined. The thread cutting starts after receiving 1-turn signal from the spindle coder, and so the cutting points on the circle of machining workpiece are the same when repeating the thread cutting many times, the spindle speed must not be changed at the same time, otherwise there is the error of thread cutting.

4) The thread machining cutting speed is controlled by P113 (max. cutting feedrate); when the above speed exceeds the max. feedrate, the system alarms. It is suggested that the thread machining cutting speed should be less than 3000 mm/min; when the speed is too big, the motor cannot response to cause the confused thread tooth.

Thread feedrate format is as follows:

inch thread speed ＝N×25.4 / E

metric thread speed＝N×P

N — speed (unit：r/min) max. speed is less than 2000r/min.

P — thread lead (unit：mm) it is switched into the metric unit to count when it is the inch thread.

5) In thread cutting start and end, the lead is not correct because of raising/reducing speed, so, the commanded thread length should be longer than the actual required length. Generally, length in the raising speed > 1.3 mm.

6) The pitch P/E is the pitch of the thread axis

【Note】

1）Start the spindle before machining the thread; otherwise, the system keeps waiting all the time (not system crash).

2）In the course of thread cutting, the feed hold key and the feedrate override are invalid; the feed stops as the spindle stops.

3）As soon as the spindle starts, do not machine the thread because of the unstable spindle speed,

otherwise, the thread machining precision may be affected.

4）In thread cutting, the photoelectric coder with 1200 or 1024 lines are generally installed to rotate with the spindle synchronously. The selected coder lines must be the same as those of the actually-installed coder. When the coder lines are 1200, P209 should be set to 1200; when the coder lines are 1024, P209 should be set to 1024. If P209 setting is wrong, the pitch will be incorrect when the thread is machined.

5）When the previous block and the current one are both the thread cutting command, the system does not detect the thread head signal (only one per revolution) but directly starts movement. This function can realize the continuous thread machining.

E.g. G33 W-20 P3; The system detects 1-turn signal in thread cutting.

G33 W-30 P2; The system does not detect 1-turn signal in thread cutting. (No empty line, comment line or other blocks are allowed between the two blocks)

6）The command must not be in the same block with other commands. It should be an independent block.

【Example】

Comprehensive example: as Fig. 4-4

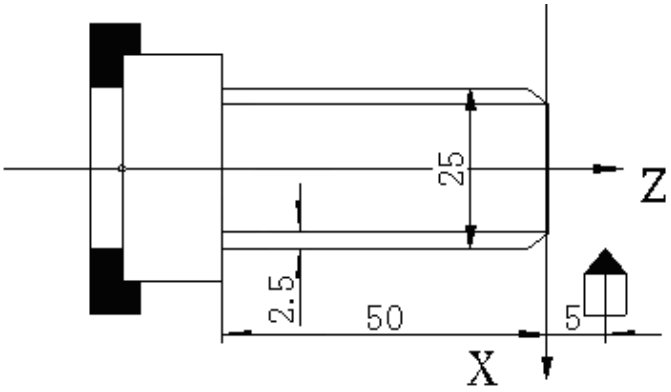


Fig. 4-4 thread cutting

Example：Thread lead: 2.5mm, cutting depth: 2.5mmas Fig. 4-4:

```
N0000  G00  X12.5  Z5                    ; approach to workpiece
N0010  G01  X11.5  F100; tool infeed 1 mm
N0020  G33  Z-50  P2.5          ; the 1st thread cutting
N0030  G00  X13                          ;
N0040  Z5                                ; return to the starting point
N0050  G01  X10                          ; the 2nd tool infeeding 1.5 mm
N0060  G33  Z-50  P2.5                   ; the 2nd thread cutting
N0070  G00  X13                          ;
N0080  Z5                                ; Z returns to the start point
N0090  M30
```

4.5.2 G34 — Variable Pitch Thread Cutting

[Command format]

G34 Z（W)_ P（E)_ Q_ R_ H_ ；Z axis variable pitch thread cutting

G34 X（U)_ P（E)_ Q_ R_ H_ ；X axis variable pitch thread cutting

G34 Y（V)_ P（E)_ Q_ R_ H_ ；Y axis variable pitch thread cutting

[Field]

P —— metric thread lead. P range:0.001mm～500.000mm

E —— inch thread lead. E range: (0.060～25400.000) tooth/inch

When P/E is the positive, it means the axial thread, and Z is the thread axis, it is the negative, it means the end face thread, and X is the thread axis.

X(U)/Z(W)/Y(V)——The absolute/relative coordinate of thread end; X, Z, Y, U, W and V range: -9999.999mm～9999.999mm; the movement value of each thread direction along Z, X and Y axes cannot set to 0.

Q —— Initial angle, which is 0°if not specified; it is used to thread part. Q range：0°～360.000°

R —— pitch incremental or decrement value per revolution of the spindle. The pitch decreases when R is negative; the pitch increases when R is positive.

R range:-500mm/pitch～+500.00mm/pitch; When R is set to 0, the pitch does not change.

H —— It means the mark of the execution method; each one means different meaning (H_d7~H_d2, H_d0: Reserved); it is 00000000 by default.

H：00000000～11111111

H_d1 = 0，pitch skip increases or reduces after the spindle rotates one turn.

H_d1 = 1，the pitch even increases or reduce after the spindle rotates some angle.

[Explanations]

Differences between G34 and G33:

1. When the previous and the current blocks are all shared with G34 command, both of them are needed to check the thread head signal at the beginning of the cutting (1/rev.).
2. Bit parameter H definitions in G34 and G33 are different.
3. G33 is used to machining the constant pitch thread and G34 to machining variable pitch thread; others are the same. Refer to G33 for details.

In G34, R is the pitch incremental value or reducing value per revolution of the spindle, R=P2-P1，R with the direction; P1>P2: the pitch decreases when R is negative; P1<P2: the pitch increases when R is positive (as Fig. 4-5).

When the pitch becomes negative or exceeds the permitted range due to the decrease/increase of R (the system will pre-count the increase/decrease path in advance), and the system alarm may occur.

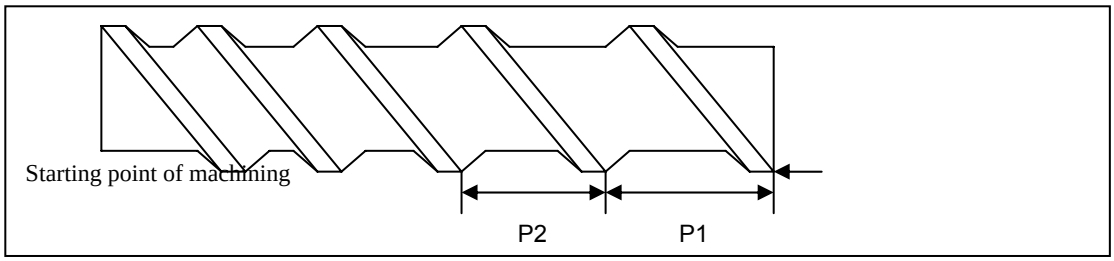


Fig. 4-5 variable pitch thread

4.6 G32 —Tapping Cycle

【Command format】

G32 Z(W) _ P(E) _ H_ ; Z axis tapping

G32 X（U)_ P（E)_ H_ ； X axis tapping

G32 Y(V) _ P(E) _ H_ ; Y tapping, only single-axis feeds

【Field】

Z(W), X（U), Y(V) ： end point coordinates of tapping or tapping length; Z /X//Y is separate X/Y absolute coordinate, W/U/Vis separate Z/X/Y relative coordinate; the user only uses one of the relative and the absolute coordinate, and the relative coordinate is relative to the movement of the current position. X, Y Z, U, V, W：-9999.999mm～9999.999mm

P：metric thread lead; P：0.001mm～500.000mm

E：inch thread lead; E：0.060 tooth/inch～25400.000 tooth/inch

H：The mark of execution method (Reservation disabled); H range：00000000～11111111

[Relative parameters]

Parameters related **to G32: P100~P108, P112, P113, and P114.**

[Explanation]

G32 Z tapping cycle execution process:

① Spindle rotation starts; Z axis infeed tapping.

② Close the spindle.

③ Wait the spindle to exactly stop.

④ The spindle rotates counterclockwise(reverse to the previous rotation direction)

⑤ Z tool retraction to the starting point of the cycle.

⑥ The spindle stops.

[Note]

1）Determine the spindle direction according to the possible tapping direction before tapping. The spindle will stop after the tapping ends. Restart the spindle when continuously machining.

2）There is a deceleration time after the spindle is closed, at the moment; Z/Y rotates along with the spindle till the spindle stops completely. Therefore, the actual bottom hole of machining is deeper than the actual required. The actual depth should be determined by the spindle speed in tapping and by whether the spindle's brake is installed or not.

3）The system forbids executing G32 when the system is in DRY RUN mode.

4）The other cautions are the same as those of G33.

【Example】

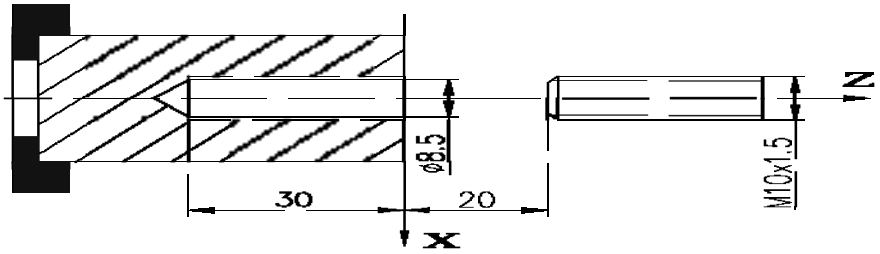


Fig. 4-6 Tapping

Example：Single thread with 1.5 mm lead

```
N0010  G00  X0  Z20          ; rapidly positioning the starting point of workpiece
N0020  M3  S01   ; spindle CW
N0030  G01  Z2  F500 ; Z approaches the workpiece
N0040  G32  Z-30  P1.5        ; Z infeed tool tapping
N0050  G00  Z20               ; leave the workpiece and return to program starting point
N0060  M02                    ; end of program
```

4.7 G38 — Rigid Tapping, Threading

[Code format]

G38 Z（W）_ P（E）_ H_ ； Z axis tapping.

G38 Y（V）_ P（E）_ H_ ； Y axis tapping, only feed with single axis.

[Field meaning]

Wherein: Z (W), Y (V): from tapping to the specified place by the command;

Z/Y is separately regarded as absolute coordinate along with the Z axis/Y axis, W/V is separately treated as the relative coordinate along with the Z axis/Y axis; one of them can be used; the relative one equals to the offset value based upon the current position.

P: Metric thread leading;

E: Inch thread leading

H: H_d0=0 means M03 tapping; H_d0=1 means M04 tapping.

[Field range]

Y, Z, V, W：-9999.999mm～9999.999mm

P：0.001mm～500.000mm

E：(0.060～25400.000) tooth/inch

H：00000000～11111111

[Explanations]

G38 The execution procedure of Z axis tapping cycle:

i. If the spindle is being rotated, and then perform the M05, the spindle will absolutely stop.

ii. The Z axis is always moved along with the spindle after entering the G38 mode.

iii. The spindle speed is gradually raised after starting the spindle, and the Z axis then moves to the object position.

iv. The spindle speed will gradually descend after the Z axis reaches to the object position till it is absolute stopped.

v. And then, the Z axis is performed the tool-retraction by the subsequent commands.

vi. The spindle starts reversely after performing the subsequent command, and the speed is gradually raised, and the Z axis moves follow with the spindle reverse direction.

vii. The spindle speed is gradually descended after the Z axis reaches to the objection position till it is absolutely stopped.

viii. Repeated again and again till to the subsequent command ends, and then retreat from the G38 mode.

[Notice]

1) The spindle will still hold a certain deceleration time after this command is closed the spindle; in this case, Z axis/Y axis will still moves depending on the rotation of the spindle, till the spindle absolutely stops. And therefore, a hole at the bottom of the screw thread should be deeper than the actual needed during actual machining, and its length should be determined based upon the spindle high/low speed and its speed variable during tapping.

2) The coordinate position cannot absolutely confirm after the G38 and subsequent commands are performed.

3) Only the separated Z (W) filed or S, M00 command can be performed in the subsequent command of the G38; the next block of the subsequent command can be followed with the subsequent command again; if the next block command of the subsequent command is other commands, and then retreat from the G38 mode.

4) Prohibit performing the G38 command in the system during the dry run.

[Example]

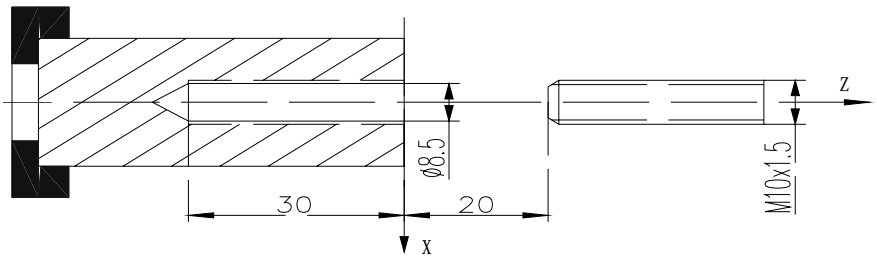


Fig. 4-7 Tapping

For example: G38 tapping, threading; screw thread leading 1.5mm.

```
N0000  G00  X0  Z20          ; Rapid positioning the workpiece start
N0010  S300                 ; The speed during tapping setting
N0020  G01  Z2  F1000       ; Z axis closes to the workpiece, reserve 2mm clearance in
                              the front of it
```

```
N0030  G38  Z-30  P1.5  H0          ; Z axis feed tapping
N0040  S500                        ; Set the speed when Z axis retracts
N0050  Z2                          ; Z axis retracts
N0060  S300                        ;
N0070  Z-30                        ; Tapping again, remove the burr
N0080  S500                        ;
N0090  Z2                          ; Z axis retracts
N0100  M02                         ; End of program
```

4.8 G50 — Setting a Workpiece Coordinate System

The system directly modifies the current tool nose coordinates into the coordinate values set by G50 in G50 and sets the current machine coordinates to the program reference point.

After the system executes G50, the front of the machine coordinates of the corresponding axis has green icon , which is taken as the program reference point return prompt.

The coordinate system created by G50 is taken as the workpiece coordinate system. After the coordinate system is created, the position of absolute coordinate in the following command is the coordinate values in the coordinate system.

X of the workpiece coordinate system is defined in the rotary center of the workpiece when the system creates the workpiece coordinate system, Z is defined in the end face of the chuck or the workpiece.

【Command format】

G50 Z_ X_ Y_; three-axis workpiece coordinate system setting

G50 Z_ X_ ; two-axis workpiece coordinate system setting, it can be Z/X, Z/Y, X/Y 2-axis

G50 X_ ; X single-axis workpiece coordinate system setting, it can be Z, X, Y single axis

G50 S_ ; max. spindle speed limit in constant surface speed. Refer to G96, G97 mode.

【Field and range】

Z, X, Y are absolute coordinate values. S limits max. spindle speed(r/min). Refer to G96, G97.

X, Z, Y：-9999.999mm～9999.999mm

【Explanation】

1) Z/X/Y cannot be shared the same block with S. Z/X/Y can select single-axis, two-axis or three-axis to set the workpiece coordinate system.

2) G50 is alone in one block and cannot be in one block with other commands.

3) Because the system has created one workpiece coordinate system and one program reference point, it uses the new workpiece coordinate system and the program reference point in Auto and Jog working mode till it is replaced after the system executes G50.

【Relative parameter】

Parameters related to G50: **P000, P001, P002**.

【Example】

G50 X100 Z100 Y100 ; creating workpiece coordinate system with three axes

4.9 G51 — Recovering Workpiece Coordinate System Setting

【Command format】

G51

【Explanation】

G51 must be specified in an independent block. It cannot be in the same block with other commands.

Recover the workpiece coordinate system and reference point set in Jog operation mode. When the workpiece coordinate and the program reference point set in Jog operation mode are replaced after G50 is executed, use G51 to recover them. After recovering, the system uses the previous workpiece coordinate system and program reference point in Jog and Auto operation modes till they are replaced.

4.10 G26 — X, Z, Y Reference Point Return

Return to the program reference point through the intermediate point at rapid traverse rate by G00.

【Command format】

G26 Z(W)_ X(U)_ Y(V)_; Z/X/Y axis rapidly traverse to the return reference position by the middle point simultaneously.

G26 Z(W) _; Z rapidly moves to the program reference point through the middle point, and other axes do not move

G26 Y(V) _; Y rapidly moves to the program reference point through the middle point, and other axes do not move

【Field and range】

X, Z, Y： Absolute coordinates of middle point;

U, W, V：Relative movement from the starting point to middle point.

X, Z, U, W, Y, V：-9999.999mm～9999.999mm

【Explanation】

1）After the command is executed, all specified coordinate axes move to the point defined by G50. They move the program reference point defined in the Jog working mode without using G50. After the corresponding axes execute the reference point return, the fronts of their machine coordinates have the blue icons as the prompts.

2）The command and other commands cannot in the same block.

3）The execution mode of the command is the same that of G00; Relative parameter is referred to G00, G50.

4）Z/X/Y can select single-axis, two-axis or three-axis simultaneously to execute the program reference point return.

【Example】

G26 Z150 X100 Y100 ; return to program reference point through the middle point (Z150，X100，Y100)

G26 U0 ; X directly returns to program reference point and other axes do not move

4.11 G28 — Return to Machine Zero(Machine Reference Point)

【Command format】

G28 Z(W) _ ; Z rapidly moves to machine zero through the middle point and other axes do not move

G28 X(U) _ ; X rapidly moves to machine zero through the middle point and other axes do not move

G28 Y(V) _ ; Y rapidly moves to machine zero through the middle point and other axes do not move

【Field and range】

X, Z, Y：Absolute coordinates of middle point;

U, W, V：Relative movement from starting point to middle point.

X, Z, U, W, Y, V：-9999.999mm～9999.999mm

【Relative parameter】

Main parameters related to G28: P021～P026, P109, P110, P111, P406 and P407.

【Explanation】

1）When the machine zero check devices(zero switch and deceleration switch included) are not installed on the corresponding coordinate axis, P406 corresponding bit parameter is set to 0; in executing G28, the system does not check the zero signal and deceleration signal, and the axis moves the middle point and returns to its zero coordinate position.

2）When the machine zero check devices are installed on the corresponding coordinate axis, P406 corresponding bit parameter is set to 1; in executing G28, the axis rapidly moves to the machine zero from the starting point and does not go through the middle point; i.e. the above machine zero return is the same that of Jog working mode.

3）The machine zero return in G28 is referred to PROGRAMMING, 4.4 Machine Zero Return in JOG working mode.

4）G28 is one short G command; its execution is the same that of G00; other relative parameters are referred to G00.

5）After the corresponding axes execute the machine zero return, there is blue icons as the prompts following machine coordinates.

6）The command and other commands cannot in the same block.

7）Z/X/Y only executes the single-axis machine zero return.

【Line-up tool】

G28 U0 **;** X rapidly moves to machine zero through the middle point and other axes do not move

G28 W0 **;** Z rapidly moves to machine zero through the middle point and other axes do not move

G28 V0 **;** Y rapidly moves to machine zero through the middle point and other axes do not move

4.12 G30 — The 2nd , the 3rd Program Reference Point Return

【Command format】

G30 P2 Z（W)_ X（U)_ Y(V)_ ; Z/X/Y axis rapidly traverse to the 2nd reference position of program return by a middle point simultaneously.

G30 P3 Z（W)_ X（U)_ Y(V)_ ; Z/X/Y axis rapidly traverse to the 3rd reference position of program return by a middle point simultaneously.

G30 P2 Z（W)_ X（U)_ ; Z/X axis rapidly traverse to the 2nd reference position of program return by a middle point simultaneously.

G30 P3 Z（W)_ X（U)_ ; Z/X axis rapidly traverse to the 3rd reference position of program return by a middle point simultaneously.

G30 P2 Z（W)_ ; Z axis returns to the 2nd reference position by a middle point, the other axes disabled.

G30 P3 Y（V)_ ; Y axis returns to the 3rd reference position by a middle point, the other axes disabled.

【Field and range】

X, Z, Y： Absolute coordinates of middle point;

U, W, V：Relative movement from starting point to middle point.

P2 specifies 2nd program reference point; P3 specifies 3rd program reference point..

X, Z, U, W, Y, V：-9999.999mm～9999.999mm

P：2 or 3

【Relative parameters】

Main parameters related to G30: **P003～P008.**

【Explanation】

1）The execution mode of the command is the same that of G00; Relative parameter is referred to G00.

2）The command and other commands cannot in the same block.

3）Z/X/Y can select single-axis, two-axis or three-axis simultaneously to execute 2nd, 3rd program reference point return.

【Example】

G30 P2 Z150 X100 ; return to 2nd program reference point through the middle point (Z150，X100)

G30 P3 Z150 X100 ; return to 3rd program reference point through the middle point (Z150，X100)

G30 P3 W0 ; Z rapidly returns to 3rd program reference point directly.

4.13 G04 — Timer Delay

【Command format】

G04 D__ ; Timer delay

【Field and range】

D — Delay time. (unit：s) G04 defines the time interval between two blocks.

D：0～9999.999 s

【Explanation】

1) G04 cannot be in the same block with other command except for S, F command.

2) The execution of the dwell time will not be affected by CYCLE START key when G04 is being performed.

【Example】

Example ： G04 D2.5 ; dwell 2.5s

4.14 G96 —Constant Surface Speed Control, G97 —Constant Surface Speed Cancel

【Command format】

G96 S ; constant surface speed control

G97 S ; cancel constant surface speed control

【Field and range】

S specifies the constant surface speed in G96. (unit: m/min)

S specifies to cancel the constant surface speed in G97.(unit: r/min)

S：0～9999 r/min

0～9999 m/min

0～4(multi-gear spindle)

【Explanation】

G96, G97 are the modal word in the same group but one of them is valid. G97 is the initial word and the system defaults G97 is valid when the system is switched on.

When the machine tool is turning it, the workpiece rotates based on the axes of spindle as the center line, the cutting point of tool cutting workpiece is a circle motion around the axes, and the instantaneous speed in the circle tangent direction is called **cutting surface speed**(for short **surface**

speed). There are different surface speed for the different workpiece and tool with different material.

When the spindle speed controlled by the analog voltage is valid, the constant surface control is valid. The spindle speed is changed along with the absolute value of X absolute coordinates of programming path in the constant speed control. If the absolute value of X absolute coordinates adds, the spindle speed reduces, and vice verse, which make the cutting surface speed as S command value. The constant speed control to cut the workpiece makes sure all smooth finish on the surface of workpiece with diameter changing.

Surface speed=spindle speed × |X| × π ÷1000 (m/min)

Spindle speed: r/min

|X|: absolute value of X absolute coordinate value, Unit: mm

π ≈3.14

In G96, the spindle speed is changed along with the absolute value of X absolute coordinates value of programming path in cutting feed (interpolation).

In G96 (constant surface speed control), Z coordinate axis of workpiece system must coincide with the axes of spindle (rotary axis of workpiece), otherwise, there is different between the actual surface speed and the defined one.

In G96, G50 S_ can limit max. spindle speed (r/min). The spindle actual speed is the limit value of max. speed when the spindle speed counted by the surface speed and X coordinates value is more than the max. spindle speed set by G50 S_. After the system is switched on, max. spindle speed limit value is **P305**(max. spindle speed of constant surface speed).

Max. spindle speed limit value defined by G50 S_ is reserved before it is defined again and its function is valid in G96. Max. spindle speed defined by G50 S_ is invalid in G97 but its limit value is reserved.

【Relative parameters】

Main parameters related to G96, G97: **P410_d6, P304**, **P305;**

P410_d6 setting determines whether the system executes the constant surface speed control in G96;

Min. and Max. speed limit **P304**, **P305** of constant surface speed control.

【Note】

1) The system issues E272- S value cannot be 0 in G50when G50 S0 is programmed.

2) The constant surface speed function can actually control the spindle speed only when the variable-frequency spindle is used, i.e. bit parameter P410_d6 = 1. If the gear-shift spindle is used, the constant surface control cannot be performed even if G96 command is specified.

3) The constant surface speed control can be performed anytime when executing **G00, G01**; the constant thread speed control is invalid during thread cutting when executing **G32, G33, G34**; when X coordinate is changed in G50 or executing the tool offset, the spindle speed adjustment is valid in the next command.

4) Min./max. speed of constant surface speed control is set by **P304/P305**respectively.

5) The rotary axis in the constant surface speed must be set in Z axis (X=0) of the workpiece coordinate.

6) G96 is modal command. When G96 is valid, the single S command is taken as the new surface speed data.

7) In DRY RUN mode, the constant surface speed control is valid. Whether the single S command is executed in Dry run is controlled by **P401_d7: P401_d7**= 0: T, M, S command is executed in DRY RUN; **P401_d7**=1: T, M, S command is not executed.

8) The constant surface speed control is invalid to get the constant spindle speed in thread cutting.

9) In G96, when the system defaults S word, the system automatically counts the surface speed as S value according to the current spindle speed(the last speed is taken as the current speed when the spindle does not start, or the spindle initial speed is taken as the current speed when the system is switched on just now) and X absolute coordinate value of tool nose.

10) In G97, when the system defaults S word, the system automatically counts the speed as S value according to the current constant surface linear speed and X absolute value of tool nose.

11) In AUTO operation mode, if the initial state of the program execution is G96, the system automatically changes G96 into G97 after CYCLE START key is pressed. The constant surface cutting is valid in machining the arc and straight line in the course of machining.

12) In the Manual working method, G96 is enabled when moving the X coordinate regardless of in feed or rapid state, the spindle speed will vary from one coordinate value to another.

13) S value of G96 is limited by S value of G50 Sxxxx and P304, P305. The S value of G50 is initialized to P305 upon power on, and their limit relationship is as follows:

- **G50 Sxxx<P304: G50 Sxxxx** is invalid, the spindle speed is adjustable between **P304~P305 in G96 .**
- **G50 Sxxx>=P304: G50 Sxxxx** is valid. The min. value of spindle speed range is limited by P304 and max. value is limited by G50 Sxxx or P305, whichever is smaller.

【Example】

P304 value is the lower limit of the spindle speed in constant surface control. When the spindle speed counted by the surface speed and X coordinate value is lower than P304 value, the lower limit of the spindle speed is the actual spindle speed.

Example：as Fig. 4-8; surface speed is 300m/min which can count the spindle speed annotated in Table 4-1.

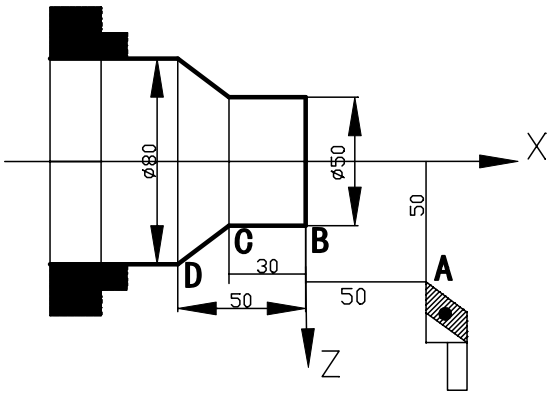


Fig. 4-8

Table 4-1

N0010	M3 G96 S300;	(Spindle rotates clockwise, the constant surface speed control is valid and the surface speed is 300 m/min)
N0020	G0 X50 Z50	(Rapidly traverse to A point with spindle speed 955 r/min)
N0030	G0 X25 Z0	(Rapidly traverse to B point with spindle speed 1910 r/min)
N0040	G1 W-30 F200;	(Cut from B to C with spindle speed 1910 r/min)
N0050	X40 W-20 F150;	(Cut from C to D with spindle speed 1910 r/min and surface speed 1194 r/min)
N0060	G0 X50 Z100;	(Rapidly retract to A point with spindle speed 955 r/min)
N0110	M30;	(End of program, spindle stopping and coolant OFF)

4.15 G22, G80 —Program Part Cycle

In the course of actual machining, for the part of workpiece or the formed parts, the part cycle command is employed to simplify the programming. The cycle body of part cycle is defined by part programming. The coordinates of end point are determined after executing this cycle.

【Code format】

G22 L_

…
…
…

Cycle body program

G80

【Field and range】

G22 defines the starting of cycle body;

L： defines the cycle times, range 1～9999;

G80 defines the end of cycle body.

【Explanation】

Command execution process：

① G22 defines the starting of cycle body and L defines the cycle times.

② Execute the cycle body program.

③ When G80 cycle body ends, if L is not 0, it reduces by 1 and then the system executes the cycle body program again; if L is 0, the cycle ends, and the system executes the following program in order.

【Note】

1）G22 and G80 must be used at the same time; G22 and G80 can be embedded;

2）The subprogram can be called in the loop body. It can have M96 and M97. Regarding as the component which shape has already determined and needs to the roughing, using G22 and G80 programming is very convenient and can improve the processing efficiency.

3）For axis needing creating the cycle offset, the program in the cycle body uses the relative programming. There is the offset between the exit coordinates and inlet coordinates of the cycle body to get the same cycle program and machining contour, and the different machining path every time.

【Example】

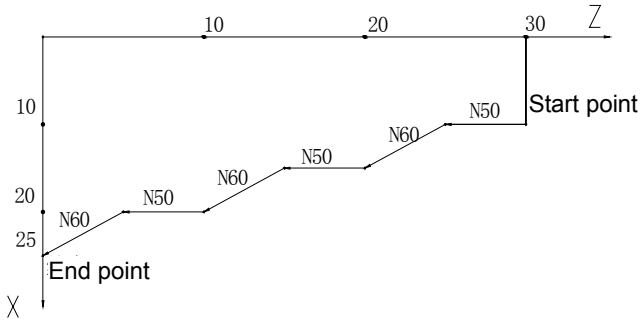


Fig. 4-9

Machining the workpiece as Fig. 4-9 cycle programming with G22, G80 as follows:

```
N0000  G50  X50  Z100      ; Define a coordinate system
N0010  M3  S01              ; Start the spindle, set to the low speed
N0020  M8                   ; Cooling ON
N0030  G00  X10  Z30        ; Rapidly position to the starting point of the cycle
N0040  G22  L3              ; Program cycles three times
N0050  G01  W-5  F50        ; Z negatively cuts 5mm，F=50mm/min
N0060  U5  W-5              ; X positively cuts 5mm，Z negatively cuts 5mm
N0070  G80                  ; End of cycle body
N0080  G26 X50  Z100        ; X, Z rapidly retracts to program reference point
N0090  M5                   ; Stop the spindle
N0100  M9                   ; Cooling OFF
N0110  M2                   ; End of program
```

The above example is programmed according to the actual shape. The programming method in Fig. 4-9 is applicable to the roughing like forging and casting, which can sharp the raw-material with high efficiency and improve the machining efficiency.

4.15. 1 G80 Adds L Address Character

The times of the program cycle can be changed by the G80 L command. L resolution: 0~9999. L=0 means that the program cycle times of the modification is 0, and then the program cycle ends. Generally, G80 L0 must be followed with end-of-program command; otherwise, the “E228: No end-of-program command” alarm will occur after calling.

[Example]

```
N0000  G50  X100  Z100          ; Define a coordinate system
N0010  M3  S01                  ; Open a spindle and then set at the low speed
N0020  G00  X10  Z30            ; Rapid position to the cycle start
N0030  G22  L5                  ; Program cycle starts, and continues for 5 times.
N0040  G01  W-5  F50
N0050  U5  W-5
N0060  if(r1005=0)  then(P0120)   ; If the No. 5 input is LOW, then return to the N0120
```

block.

```
N0070  G80                          ; The end of cycle body
N0080  G26  X100  Z100              ; X, Z rapid retracts to the program reference point
N0090  M5
N0110  M2
N0120  G80  L3                      ; Modify the cycle times: 3 times
```

[Notice]

G80 L and G22 cannot be returned at the same exit during the program is performed; otherwise, the limitless cycle may occur. Refer to the following:

G22 L＊＊

……

G80 L＊＊

4.16 G98 —Feed per Minute, G99 —Feed per Revolution

【Command format】

G98 **F**＊＊＊＊＊．＊＊＊ ; feed per minute

G99 **F**＊＊＊＊＊．＊＊＊ ; feed per rev

【Field and range】

G98 F＊＊＊＊＊．＊＊＊ ; F unit：mm/min;

F －define the feedrate of its following interpolation command, its unit: mm/min.

G99 F＊＊＊＊＊．＊＊＊ ; F unit：mm/r;

F － define the feedrate of its following interpolation command, its unit: mm/r.

F：0～15000.000mm/min

0～15000.000mm/r

【Note】

1) The value format of G98/G99 must be matched with that of F word. Otherwise, the system issues：Missing information.

2) G98/G99 is the modal command and it keeps valid before being changed. The single F command can be taken as the new feedrate.

3) G98 is the initial state of the system and the system automatically sets to G98 during initialization (mm/min).

4) When F=0, the system suspends and prompts an alarm message when executing the machining the program.

5) In G99, when the spindle speed is 0, the feed automatically pauses and the system displays “Pause: the spindle speed is 0”. It keeps the state till the spindle speed is not zero; then, by pressing CYCLE START key again, the system recovers the execution.

6) In G99, generally, when the spindle is just started, the system should delay a proper period of time to wait the spindle speed to become stable, and then executes the cutting. Otherwise, the

system fails to detect the spindle speed, displaying “Pause: the spindle speed is 0”.

【Example】

```
…
N0100   G98 F800        ; define the feed per minute，F feedrate: 800 mm/min;
…
N0160   F50             ; F feedrate: 50 mm/min;
…
N0200   G99 F2.1        ; set the feed per rev，F feedrate: 2.1 mm/rev;
…
N0250   F0.56           ; F feedrate: 0.56 mm/rev.
…
…
```

4.17 G31 — Skip

The execution of G31 is the same as that of G01, and the difference is that G31 constantly check the external input interface signal (G31I) state. When the signal meets the condition, the coordinate axis stops the feed and continuously executes the next command, otherwise, the coordinate axis feeds to the end point, and then the system goes to the next command.

【Command format】

G31 Z（W)_ X（U)_ Y（V)_ F_ H_ ；Z/X/Y 3-axis feed simultaneously

G31 Z(W) _ X(U) _ F_ H_ ; Z/X two-axis feed simultaneously, or Z/X, Z/Y, X/Y two-axis feed simultaneously

G31 Y(V) _ F_ H_ ; Y single-axis feeds

【Field and range】

X, Z, Y： End point coordinates after the command ends.

U, W, V： Incremental value counted by the theory end point.

F：Feedrate. F is modal and can be omitted when it is not changed; it is controlled by the feedrate override.

H：The mark of execution method, each one means different meaning (H_d7~H_d1: Reserved) it is 00000000 by default.

H_d0=0: G31I LOW meets the skip condition;

H_d0=1: G31I HIGH meets the skip condition;

X, Z, Y, U, W, V：(-9999.999～9999.999) mm

F：0.001mm/min～15000mm/min

H:00000000～11111111

【Explanation】

Z/X/Y can execute single-axis, two-axis or three-axis feed simultaneously.

G31 traverse speed is specified by F and controlled by the feedrate override.

Actual feedrate =F x feedrate override

G31 is non-modal command, and can be omitted in the next block when it is the same.

【Note】

1）G31 input check function is non-standard pin input control; when the function is needed, the user can define its pin in the interface parameter P532(G31**I**) and correctly connects with the wirings. The concrete interface parameter definitions are referred to **OPERATION, 4.6 Parameter Working Mode**; the concrete wiring connection is referred to **CONNECTION, Chapter 3 CNC Device Connection**.

2）To get the exact position, the run decelerates once the system has checked the external input interface signal change. So, the actual feedrate in G31 is not too high and it is suggested that it should be less than 1000mm/min, otherwise, G31 cannot realize the skip function because it cannot check the interface signal.

【Relative parameter】

Interface parameter P532 defines the input interface pin.

The relative parameter of cutting feed is referred to G01.

【Example】

Current: X=100 Z=100;

G31 Z30 F500;

The move distance is Z70, Z moves at 500mm/min continuously to Z30 when G31I signal cannot meet the skip condition; Z＝68.37: G31**I** meets the skip condition and Z immediately stops running and the system executes the next command.

4.18 G35 — I/O Signal Observation Command

The specified parameter block can be compiled by G35 command in the overall programs.

[Code format]

G35 I1.5.9…32 K1.3.6…32

[Filed meaning]

I: Input port signal. There are 32 numbers (1-32) can be followed with the I, and the numbers are separated by “.”;

K: Output port signal. There are 32 numbers (1-32) can be followed with the K, and the numbers are separated by “.”.

[Explanation]

The I/O signal observation interface is appeared by “LEFT” key in the Manual or Auto working mode after this command is performed. The function state of input/output ports signal followed with I/K can be observed in this interface. If there are multiple pages will be displayed in the window, and the I/O signal function state can be checked by UP/DOWN page key.

[Notice]

In a same block, simultaneously, if there are some self-defined commands, such as G35, M50~M74 are performed, and then, display the priority level in the Manual/Auto mode: G35 defines the top one; then the M50-M74; finally, the command commands M, T, S.

[Example]

```
N0000 G50 X100 Z150
N0010 M10
N0020 M03 S500
N0030 M08
N0040 T11
N0050 G35 I8.12 K17
N0060 G00 X65 Z103
N0070 G01 X0 F60
N0080 M81 I8.0      ; Observe the input/output pin state when displaying the window after controlling
                      the "LEFT" key. Perform the next one if the input UI08 is LOW, otherwise, keep
                      waiting.
N0090 M82 Q17.0 I12.0 D3    ; Make the No.UO17 pin outputting the LOW, the longest hold time D,
                              retract the output; The No.UI12 pin is LOW, then perform the next
                              one.
N0100 G00 X100 Z100
N0110 M09
N0120 M05
N0130 M02
```

When the above-mentioned programs are performed, at the beginning, the common commands M, S, T can be displayed in the window by pressing the "LEFT" key; the U I08, UI12 and UO17 pins change can be displayed in this window after performing the G35 command.

4.19 G52 — Rotary Axis Y Axis Coordinate Clearing

When the controllable axis is used to control the rotary axis, in MANUAL or AUTO mode, the system can use G52 to clear the integer and keep the remainder.

In G52, when "absolute values of tool nose coordinate of the current controllable axis ≥the integrated value set by G52, the system clears the integer coordinate value and leaves the remainder. The machine coordinate of controllable axis also clears the corresponding coordinate value.

【Command format】

G52 Y(V)_ ; Y axis coordinate integer

G52 Z（W)_ ； Z axis coordinate integer

G52 X（U)_ ； X axis coordinate integer

【Field meaning】

Y（V）：The integer radix of the Y axis；

Z（W）：The integer radix of the Z axis；

X（U）：The integer radix of the X axis.

[Field range]

Y, V, Z, W, X, U：-9999.999～9999.999 (Without meaning of numerical symbol)

【Explanation】

1) When **P027~P029**is set to 0, it means that Y is not the rotary axis.

2) When the system parameter sets Y(**P405_d1**=1)，G52 Y is valid.

3) When the X axis is set in the system parameter (**P405_d0=1)**, the G52X command is then enabled.

4) G52 only can be performed a block alone, which cannot be shared a same block with other commands simultaneously.

5) When the coordinate value followed with G52 is set to 0, it clears to the integrated number based upon 360..

6) There is no data field following G52, the system defaults to be G52 Y360.000.

【Relative parameter】

Parameter related to G52: **P027~P029, P405_d1, P405_d0.**

【Example】

```
G00   Y20
G01   Y136.6
G52   Y100        ; rotary axis coordinate clearing the integer 100, Y coordinate value is 36.6
```

4.20 G66—Store the Current Coordinates, G67—Return to the Stored Coordinates

【Command format】

```
G66                    ; To store the current coordinate of tool nose on Z, X, Y axes;
G67  W0  U0  V0        ; Return to the G66 position. That the axis does not specify in G67 is
                         not moved.
```

【Instructions】

X, Z, Y: If they are specified in absolute coordinates, it means the axes should rapidly traverse to those points.

U, W, V: If the relative coordinates is not 0, it means the axes rapidly traverse to the relative new coordinates.

Z/X/Y can perform rapid traverse, single axis, two-axis or three-axis movement. Absolute coordinates and relative coordinates cannot be used at the same time. The axis that does not move can be omitted, but G67 cannot be specified alone.

G66 is to store the current tool nose coordinates; G67 is to command the axis to move to the coordinates.

G67 should be used only when G66 has been used once. When G66 is used several times, the latest coordinate is stored.

G66, G67 can only be specified without other commands. The movement path of G67 is the same as G00.

【Example】

```
N0010  G00 Z350 X280 Y600; Define the machining original point and position to this point;
N0010  T11
N0100  G00 Z102 X120          ;
N0200  G66                    ; Z102,  X120  Store the current coordinates;
N0200  T22                    ; The tool nose coordinate is uncertain after tool changing;
N0300  G67  X120  W10         ; X axis moves to 120, and Z axis move to 102+10.
N0200  M02
```

4.21 G81 — Drilling；G83 — Peck Drilling

【Common format】

G81 Z（W） R D F L ；Drilling，can be Z, X or Y

G83 Z（W） R Q D F L ；Peck drilling，can be Z, X or Y

【Field and range】

Z （W）、X（U）、Y（V）：hole bottom position；range：（-9999.999～9999.999）mm

R：Distance from initial point to R point，R＜|current coordinate－hole bottom coordinate| ; range：(0～999.999)mm

D: The delay time the tool stays at the hole bottom; range (0～99.999)s

F：feedrate；range:（0.001～15000）mm/min

L：Number of repeated；range：1～9999

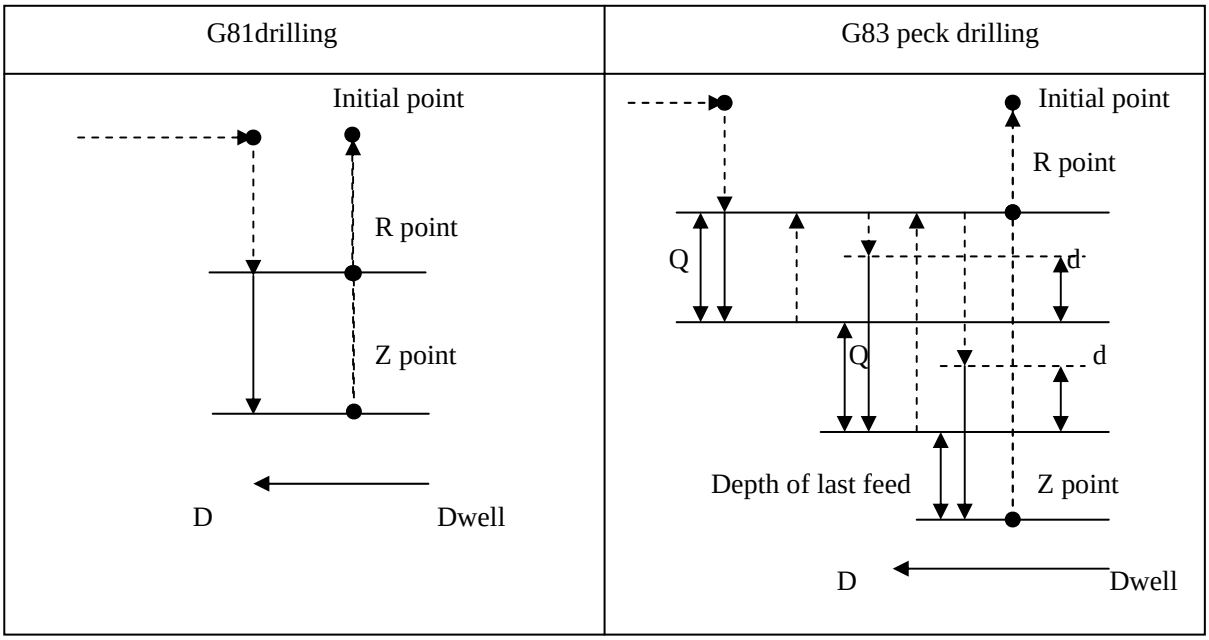
Q：each feed depth; range：（0.001～999.999）mm

【Explanation】

Z/X/Y can select any single axis.

Each tool retracting point in G81, G83 canned cycle is the initial point; G81 and G83 are non-modal commands.

Q in G83 indicates each cutting amount. Before the second cutting, the rapid feed turns to cutting feed in front of d distance (broken-line arrow in the figure). The value of d is set by P335. The execution process of G81 and G83 commands is as follows.



4.22 G0681, G0683 —The Drilling Cycle in the G06 Mode

[Code format]

G0681 Y（V） R D F L ; Drilling, Z, X, Y single axis allowed

G0683 Y（V） R Q D F L ; Peck drilling, Z, X, Y single axis allowed

[Explanation]

The drilling cycles both the G0681, G0683 and G81, G83 are identical, which are the drilling cycle in the G06 mode; it is not occupied the time during performing. Refer to the G81, G83 command for details.

4.23 Appendix: G function and Its Explanation Table(table 4-2)

Code	Programming format and explanation	Function	Group	Remark
G00	G00 X(U) Z(W) Y（V） (single, 2 or 3 axes)	Rapid traverse	1	Initial G
G01	G01 X(U) Z(W) Y（V） F (single, 2 or 3 axes)	Linear interpolation		Modal G
G06	Can only be X, Y or Z single axis, manual input is executable G06 Z（W） I8.0 F ; stops automatically after finishing the distance G06 Z+ I8.0 F ; Positive moves, soft/hard limit stop alarm G06 Z- I8.0 F ; Negatively moves, soft/hard limit emergency stop alarm G06 Reset can continue execution, stops at emergency stop; the same for X/Y axis. I8.0 ; If the low level of UI08 pin is detected, the execution automatically stops. I8.1 ; If the high level of UI08 pin is detected, the execution automatically stops.	G06 motion mode	2	One-shot G command
G07	G07 Z ; Z automatically stops; the same for X/Y axis G07 ; Without axis number, which means all stops automatically.	G06 automatically stops		One-shot G command

G08	G08 X1 Z1 Y1 ; X, Z,Y are in G6 mode The “1”behind the axis number means the axis is in G06 mode, and “0”means G6 is forbidden.	Enable/disable G06 mode		One-shot G command
G09	G09 Y0 ; Allow to move the Y axis in Manual and programming, instead of performing G6 in monitoring. G09 Y1 ; Prohibit to move Y axis in the Manual and programming, allow performing G6 in monitoring	Y axis working state shift		One-shot G command
G32	G32 Z（W） P（E） H (Z, X, Y single-axis allowed)	Tapping cycle	3	One-shot G command
G33	G33 Z（W） P（E） Q H （can be X, Z and Y axis）	Constant thread cutting		
G34	G34 Z（W） P（E） Q R H （can be X, Z and Y axis）	Variable pitch thread		
G38	G38 Z（W）_ P（E）_ H_ (Z, Y single-axis allowed) G38 Y（V）_ P（E）_ H_ ；Y axis tapping	Rigid tapping, threading		One-shot G command
G26	G26 X（U） Z（W） Y（V） (single, 2 or 3-axis) Cross the middle point and return to the program reference point; rapid traverse in G00.	Program reference point return		
G28	G28 X(U) or G28 Z(W) or G28 Y(V)	Machine zero return		One-shot G command
G30	G30 P2 X（U） Z（W） Y（V） (single, 2 or 3-axis) G30 P3 X（U） Z（W） Y（V） (single, 2 or 3-axis)	2^{nd}, 3^{rd} program reference point return		
G50	G50 X Z Y (single, 2 or 3-axis)	Workpiece coordinate system setting		
G51	G51	Workpiece coordinate system recovery		
G52	G52 Y(V)	Rotary axis coordinate clearing integer		One-shot G command
G31	G31 X（U）Z（W）F H（X、Y、Z single or 2-axis）	Skip function		
G35	G35 I K	I/O signal display		
G22	G22 L	Local cycle start	4	
G80	G80	Local cycle end		
G04	G04 D	Delay	5	One-shot G command
G66, G67	G66	Memorizing current coordinate point	6	One-shot G command
	G67 W0 U0 V0 (single, 2 or 3-axis)	Return to coordinate point memorized by G66		
G81	G81 Z（W） R D F L ((X, Y or Z single axis)	Drilling	7	One-shot G command
G83	G83 Z（W） R Q D F L ((X, Y or Z single axis)	Peck drilling		
G0681	G0681 Y（V） R D F L（Y axis）	Drilling in the G06 mode		One-shot G command
G0683	G0683 Y（V） R Q D F L（Y axis）	Peck drilling in the G06 mode		One-shot G command
G96	G96 S	Constant surface speed ON	8	Modal G
G97	G97 S	Constant surface speed OFF		Initial G
G98	G98 F	Feed per minute	9	Initial G
G99	G99 F	Feed per revolution		Modal G

Note: the system is in G00，G97，G98 when it is switched on and reset.

4.24 Appendix：

G Code and its Relative Parameter Explanation (Table 4-3)

Word	Function	Relative parameter explanation	Remark
G00	Rapid traverse	P100～P108, P112, P114, P400_d3	
G01	Linear interpolation	P112, P113, P114, P401_d5, P401_d4	
G06	G06 motion mode	P419_d0 ～ P419_d3, P402_d7, Refer to G00 for others	
G07	G06 automatic stop		
G08	Enable/disable G06		
G09	Y axis working state shift		
G32	Tapping cycle	P100~P108, P112, P113, P114	
G33	Constant thread cutting	P100~P108, P113, P116, P117, P209, P306, P307, P403_d0	
G34	Variable pitch thread	The same as G33	
G38	Rigid tapping, threading	Same to G32	
G26	Program reference point return	Refer to G00, G50	
G28	Machine zero return	P021～P026, P109, P110, P111, P406 and P407, Refer to G00 for others	
G30	2^{nd}, 3^{rd} program reference point return	P003～P008. Refer to G00 for others	
G50	Workpiece coordinate system setting	P000, P001, P002	
G51	Workpiece coordinate system recovery		
G52	Rotary axis coordinate clearing	P027, P405_d1	
G22	Local cycle start		
G80	Local cycle end		
G04	Delay		
G66	Memorizing current coordinate point		
G67	Return to the coordinate point memorized by G66		
G81	Drilling		
G83	Peck drilling	P335	
G0681	Drilling in the G06 mode	Refer to G81 and G06	
G0683	Peck drilling in the G06 mode	Refer to G83 and G06	
G96	Constant surface speed ON	P410_d6, P304, P305	
G97	Constant surface speed OFF		
G98	Feed per minute		
G99	Feed per revolution		
G31	Skip function	P532, refer to G01 for others	
G35	I/O signal display		

CHAPTER FIVE GENERAL PROGRAMMING RULES AND EXAMPLES

5.1 General Programming Rules

The command compiling must meet the rules in one block, which can be convenient to the system identifying them; it is suggested that the command character should be in the fore and the data field should be in the later. The system executes the Programming Rule Check, and the program meeting the programming rules can be compiled; alarm will occur when there is wrong.

1) There is no repetitive command in one block, otherwise, the system alarms “E202: Repetitive command”;

Mistaken example：N0200 G00 G00 Z30;

2) The command character must be followed by the valid digital command, otherwise, the system alarms “E201, Illegal command”;

Mistaken example： N0200 G23 Z30 ; no the command.

3) There must not be the repetitive data field in one block, otherwise, the system alarms “E234: the data field is repetitive” ;

Mistaken example：N0200 G00 Z20 Z30;

4) There must not be the mutually contradictory data field, otherwise the system alarms “E210; the excessive fields”;

Mistaken example：N0200 G00 Z20 W30;

5) The command character and field character must be followed by the valid digit without the blank space, otherwise the system alarms “E204: the command format error”;

Mistaken example：N0200 G00 Z 20 ;

N0200 G 00 Z30 ; there is no blank space between G and 00

6) The required data in the block cannot be omitted, otherwise, the system alarms “E206: leakage message”;

Mistaken example：N0200 G90 X100 ;

7) There must not be the fields, letters and digits unrelated to the command, otherwise, the system alarms “E203: illegal message”;

Mistaken example：N0200 G00 X W 100 ; surplus character W

8) The number of the data field must be in the valid range, otherwise, the system alarms “E211: the data exceeds the range”

Mistaken example：N0200 G00 X100 Z99999 ; the data 9999 exceeds the range.

9) It can be omitted when the first digit is zero in the command.

Example: G00 can be written to G0.

5.2 Programming Rules for Commands in One Block

Programming rules for commands in one block mean that there are many commands in one block simultaneously, but all commands are not in the same block. The system executes Programming rule check, and the programs meeting the programming rules can pass the compiling; the system alarms and prompts Some command need to be used independently. And the rules are as follows:

1) Commands only in single block:

G50, G51, G52, G26, G28, G30, G31, G35；

G32, G33, G34, G38;

G04;

G06, G07, G08, G09；

Program local cycle: G22, G80;

G66, G67；

G81, G83, G0681, G0683；

M00, M02, M20, M30, M86, M95, M96, M97, M98, M99；

M21, M22, M23, M24；

M91, M92, M93, M94；

M47, M48；

M81, M82, M83, M84, M50～M74 (The self-defined command can be shared a same block with the F, S command).

2) The system cannot judge them when some commands have mutually contradictory operations and the same data. To avoid the above, M, G commands which cannot be in the same block are divided into many groups and the commands in the different group can be in the same block as follows:

M commands which can be in the same block are divided into many groups as follows:

Group 1：M03, M04, M05;

Group 2：M08, M09;

Group 3：M10, M11, M12;

Group 4：M78, M79, M80;

Group 5：M32, M33;

Group 6：M41, M42, M43, M44.

G commands which can be in the same block are divided into many groups as follows:

Group 1：G00, G01

Group 3：G96, G97;

Group 3：G98, G99.

【Notice】

1) The commands which can be in the alone block cannot be in the same block with other commands, otherwise, the system alarms "**E214：One command needs to be separately used**", and they do not meet the rules for the commands in the same block;

2) The commands in the same group cannot be in the same block, otherwise, the system alarms

"E205: there are not compatible commands";

3) T command automatically calls M60 when the tool type is 9, so, T command is only in the alone block, otherwise, the system alarms.

5.3 Command Execution Sequence

【Relative parameters】

P401 Efficiency setting (3-level)

P401_d3： Execution sequence of many commands in the same block: 0-sublevel 1-synchronism

P401_d2：0-close 1-open rapid skip execution function:

◆ Substep execution for many commands in the same block

P401_d3=0，sublevel; at the moment, **P401_d2** is not active.

When many commands in the same block are executed, their execution sequences are as follows: after one functional command is executed, the system executes the next one till all are performed.

Table 5-1：Command execution sequence table

The execution sequence of the command (From up to down)	Function
G96/G97 S function	
S function	Spindle shift or rotation speed
G98/G99 F function	
M32	Lubrication ON
M10	Chuck clamping
M78	Tailstock forward
M41、M42、M43、M44	Spindle changes the 1st, 2nd, 3rd, 4th shift
M03、M04	Spindle open
M08	Close the coolant
T function	Tool-change
G01	Cutting command
M05	Spindle close
M09	Close the coolant
M33	Lubrication OFF
M79	Tailstock backward
M11	Chuck releasing
M12	
M80	
G00	Rapid positioning command

【Example】

N1000 T11 M03 M10 G00 X50 M08

The execution sequence of sub-step: M10, M03, M08, T11, G00 ;

All used time for executing the whole block is equal to the sum of single command execution time.

◆ Synchronous execution for many commands in the same block

P401_d3=1，synchronism; **P401_d2**=0: forbid rapidly jumping block to execution

In the synchronous execution, Refer to the table 5-1:

If there is the cutting command in this block, then the M, S, T commands before the cutting command will simultaneously performed; after it is completed, the cutting command is then performed, and then simultaneously execute the following M, S, T commands.

If the G00 command in this block, then the M, S, T command before the G00 command will simultaneously perform with the G00 command.

The next block can be performed until the overall commands are completed.

The commands which are not relative each other are almost executed simultaneously, are not executed orderly. The system automatically orderly executes M command which is interlock according to the parameters.

【Example】

N1000 T11 M03 M10 G00 X50 M08

When the parameter setting can be opened the spindle as long as it firstly clamps (**P402_d5**=0), the system follows the interlock relationship and its execution process is as follows:

1) Simultaneously start executing T11/ G00 X50/M10/M08;

2) Execute M03 after M10 is executed;

3) Execute the next block after all are executed.

When the parameter setting can be opened without clamping firstly (**P402_d5**=1), the execution process is as follows:

1) Simultaneously start executing T11/ G00/M10/M08/ M03;

2) Execute the next block after all are executed.

All used time for executing the whole block is equal to the longest single execution time.

◆ Synchronous execution for many commands in the same block and rapidly jumping block to execution

P401_d3=1，synchronism; **P401_d2**=1，rapidly jumping block to execution

In the mode, the system rapidly skips to the next block after the axis motion commands of the current block are (do not wait other M, S, T having been executed).

【Note】

1) The system follows the rule: wait the others to be executed when other commands in the same group are executed.

2) The system meets “The command only in an alone block” , it waits to execute it after the previous is executed; and it executes the next block after “The command only in an alone block” is executed.

3) When the user needs to execute the next after the previous all commands are executed, he needs to insert one G04 D0 command.

4) When the rapid trans-block executes, specially note that the distance between the tool-change and workpiece should be adequate during programming, prevent from impacting the workpiece during tool-change.

CHAPTER SIX ALARM MESSAGE

The overall warning information in this CNC system are simply prompted in the screen by Chinese characters, making corresponding disposal according to the prompting content. Each alarm meaning is shown in the following table. “Workpiece program” is also named as “machining program” in the following table.

The alarm number starts by uppercase letter E and the following is three numbers. The classification is as follows:

E001~E009: Alarm in PARAMETER, OFFSET working mode;

E100~ E199：Alarm in EDIT working mode;

E200~ E299：Alarm in PROGRAM COMMAND;

E600~ E699：Alarm in PROGRAM CHECK;

E300~ E399：Alarm in JOG, AUTO working mode executing relative operation;

E400~ E499：Related Alarm in JOG, AUTO working mode executing relative statement.

6.1 Emergency Alarm

The stop operation in system emergency, please refer the details in **OPERATION, 4.1.4.3 Emergency Alarm**.

6.2 Alarm Table in PARAMETER, OFFSET Working Mode(i.e. E001~E009)

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E001	Not support USB	USB device isn’t supported by this system software	Use USB device supported by the system
E002	Remove USB suddenly	Device is removed suddenly in making USB device check	Wrong operation, forbid this operation
E003	No insert USB	Don’t insert USB device	Insert USB device
E004	Fail to find valid content or file	USB device doesn’t create the specified catalogue and file	Create the valid catalogue and file in USB device
E005	USB save file fail	USB saving failure	Save again or check USB device
E006	No enough memory in USB	No enough U disc rest room	Change the bigger U disc or check U disc to release more space
E007	Create content fail	U disc failure in creating the specified catalogue	Check USB device and create the appointed catalogue again
E008	Create file fail	Failure in creating file	Check USB device or create the specified format file again
E009	USB read file fail	USB is failed in reading file	Check the USB device or read file again
E010	Fail to find dedicated content or file	USB is failed in reading catalogue list	Check the USB device or create this catalogue again
E011	File is too big	Too large file	File is too large, modify the file
E012	Fail to open file	Can’t open the file in specified catalogue or file is destroyed	Check the file name ,suffix if they can match the rule or check the USB equipment, create the file again
E013	No dedicated file	The file in specified list does not exist or has been destroyed	Create the file in specified file

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E014	Invalid Para NO.	Parameter number exceeds the allowed range	Change the parameter number into the available parameter number range
E015	Para NO not standard	Invalid character in parameter number, does not meet the standard	Delete the invalid character in the parameter number
E016	Specify Para NO before data	Must have parameter number in the first parameter	Add the parameter number of the first parameter
E017	Error in Para data	Parameter data exceeds the system specified range	Modify the parameter data
E018	Repeatedly input Para NO	Parameter number can't be input repeatedly	Check and delete the repeat parameter number
E019	Invalid characters in the data	Can't have illegal character in file	Delete the illegal character
E020	Wrong Para file formula	Parameter file format can't meet the requirement	Create the parameter file according to the parameter file format or add the parameter file sign
E021	S-upgrade code proof error	Software promoting command checking error	Contact the supplier
E023	Illegal I/O setting Pin NO. out of range	Interface parameter setting exceeds the maximal I/O pin number	Modify the I/O to the specified range
E024	Illegal I/O setting Input pin NO.occupied	Parameter setting and I/O input pin collision	Release this I/O or use other I/O pin
E025	Illegal I/O setting Output pin NO.occupied	Parameter setting and I/O input pin collision	Release this I/O or use other I/O pin
E030	Fail to read Para, initialization	Failure in SRAM saving parameter, can't read	Save again
E031	Illegal character OFT.	Illegal character in offset number of tool compensation transmission file	Delete the illegal character
E032	Tool compensation NO. not standard	There is no tool compensation number behind the T of tool compensation transmission file, or the offset number lack T sign	Completing the number behind the T sign in file, or add the sign of offset number
E034	OFT. NO. exceed 1—64	The tool offset number in tool compensation transmission file exceeds (1~64) range	Modify or delete the exceeded tool compensation number, change them to the specified range
E035	Illegal data	Tool compensation data in tool compensation transmission file has illegal character in	Delete the illegal character
E036	Input data error	The tool compensation input data in tool compensation transmission file has mistake	Check data according the tool compensation communication file format and correct it
E037	File symbol error	File type mark in tool compensation transition file is wrong	Modify the file type according to the tool compensation file format
E038	Send OFT fail	sending tool compensation file failure or be manually cancelled	Sending tool compensation file according to the correct tool compensation file format and operation method
E039	Receive OFT fail	Fail in receiving tool compensation file or delete manual	Receive the tool compensation file according the correct tool compensation file format and operation mode
E040	File is too big	The length of received file is out of the maximal length of buffer zone	Modify the file according the file format, correct the received file according the requirement
E044 ~ E046	Ratio of magnify & coefficient exceed (1/128-128)	Ratio between multiplication and division is out of (1/128～128) range	Modify parameter: multiplication or division

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E047	Para set conflict, input port occupied	The corresponding pin in function is occupied	Release the occupied input pin in interface parameter, or set other free input pin
E048	Para set conflict, output port occupied	The corresponding pin in inputting function is occupied	Release the occupied output pin in interface parameter, or set other free output pin
E049	Abnormal, ESC	In reading the USB process, it is disordered	Operate the USB device over again
E050	Not support USB file formula	The system only reads FAT16 or FAT32 file format	Check or change USB device
E051	USB memory is too big	The system only supports the maximal 8G USB device	Change the USB device
E053	USB fails to transmit, reset	Failure in USB data transmission process	Press system reset key and try again, or operate the USB device again

6.3 General Chart of Alarm in Edit Working Mode(i.e. E100~ E199)

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E100	Program which will delete doesn't exist	The program list hasn't the program to be deleted	wrong operation
E101	Block is too long in received program	There are excessive long block during sending the workpiece program, it is more than 250 characters.	Modify the program to be sent
E102	Program which will send doesn't exist	The program catalogue hasn't the program to be sent	Rebuilt the program to be sent
E103	No specified program	Don't choose program in program catalogue	Choose the needed program in program catalogue
E111	Received program NO. is wrong	The program number range doesn't in (0~254)	Check the receiving program's number
E112	Received program is void	No content in program	Check and modify the program
E120	Not support USB	The system can't identify this USB device	Use the USD equipment which can be identified by the system
E121	USB is removed	The system is identifying the USB equipment, the equipment is moved suddenly	wrong operation, forbidden these operation
E122	Not insert USB	Don't insert the USB equipment	Insert the USB device
E123	Stop opening USB	Manual operation, press emergency or reset key	Manual operation, stop to open the USB device
E124	Fail to save program in USB	The USB is failed in saving program	Check the USB device
E126	Surplus space in USB is not enough	The U disc doesn't have enough rest room	Change the bigger U disc or tidy U disc to release more room
E127	Contents fail	The U disc creates specified catalogue failure	Check the USB device
E128	Fail to set specified content	The program in specified catalogue can't be opened	Check the USB device or the program is destroyed and must to be created over again
E129	Fail to set program	The U disc is failed in creating program	Check the USB device
E130	Stop sending program	Manual operation, press emergency or reset key	Manual operation, exit the sending
E131	No specified content in USB:C001PRO	The USB device doesn't have the specified catalogue: C001PRO	Create this catalogue
E132	USB fails to transmit & to send now	The USB device communication error	Press reset key and then insert the USB device again
E133	Stop receiving files	Manual operation, press emergency or reset key	Manual operation, exit the receiving

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E134	Fail to read program in USB	The USB is failed in reading program	Check the USB device or create this program
E136	Program is too big	The communication program capacity is too large	The program is too large, modify the program
E137	Fail to open program	Can't open program, maybe it's destroyed	Check the USB device or create this program
E142	Program is too long, delete extra part	Program error	Check the program error again
E143	Program is too long, delete it	Program error	Check the program error again
E144	Current program is void, so can't rename	The current program is empty and can't change the name	Edit the program first
E145	USB-program to receive doesn't exist	The USB receiving program does not exist	No this program in the USB equipment, create this program again
E147	Not support file system in USB, use FAT32	The U disc isn't FAT16 or FAT32 file system	Suggest using FAT32 file system
E160	Program NO. is wrong	Only (000～254) program can be input	Modify the input program number
E161	Copied program exist	The program to be copied has existed	The program has existed, this operation can't go on
E162	Program is void & can't copy	The current program is empty and can't be copied	Edit the program first
E163	Editing 253 program exceed memory	The 4M program editing storage room is full	Can't go on the next edit, except the editing program can release some storage room
E164	Exceed program memory, fail to save	Power-off protection program storage area is full and can't protect any more	Delete some programs in system
E165	No deleting all programs	The system parameter enactment can't delete all program	Modify the enactment parameter according to the privilege
E166	Renamed program exist	The program to be changed the name has been existed, can't change the name	Please delete the old program firstly
E167	Fail to erase FLASH	System hardware: storage CMOS chip has problem	Contact the supplier, change the storage CMOS chip
E168	Fail to write FLASH	System hardware: storage CMOS chip has problem	Contact the supplier, change the storage CMOS chip
E169	Editing program exceeds memory	The storage room of 800k program editing area is full	Can't go on the next edit, except the editing program can release some storage room
E170	Fail to read solidified section %254 (no programs or destroyed)	(there are no programs in solidified section and SRAM cannot write	Reedit the program
E171	No command in help	Command input error, or don't support this command	Check the command to be looking for, input the correct command again
E172	Program line too long	Edit and replace error	Check the replacing content again
E173	Replace failed, no content replace	Edit and replace error	Check the replacing content again
E174	Processing program locked, no rewriting	Set the bit parameter P416_d0 to be 1	Set the bit parameter P416_d0 to be 0
E175	Para set no line number, no arrange again	Set the P333 to be 0, the system compositor again function is invalid	Set the P333 to be non zero value again

6.4 Emergency Alarm Program Alarm Table (i.e.E200~ E299, E600~ E699)

The type of alarm in program is divided to: Alarm in PROGRAM COMMAND, Alarm in

PROGRAM CHECK two kinds.

6.4.1 Alarm in Program Command (i.e. E200~299)

Alarm in program command, means there is wrong command in working program and then alarm, it can be eliminated by inputting the correct command, and it has little relation to the parameter.

Alarm List in programming command

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E201	Illegal command	The system undefined command appears in block	Input again according system command list
E202	Command repeat	The same command is input repeatedly in the program	Delete the repeated command
E203	Illegal information	The system unidentified command appears in block	Delete the wrong command, Input the correct command
E204	Command format wrong	The command format is wrong in the block	Input again according the correct command format
E205	Command isn't compatible	Two or more commands incompatibility appear in the same block	Delete the unwanted command or line input
E206	Lack of information	Missed the command needed content in the block	Input the command needed content according the correct format
E208	Brackets is miss	Without brackets in the statement	Brackets are added to the source program
E209	Too many commands & characters	The command character in same block is over 20	Modify source program, or line input
E210	Surplus characters	Unwanted field is input in the block	Delete the unwanted field in the block
E211	Data exceeding range	The value in the block is over the specified range	Modify the is wrong data again
E212	Errors in data	The data don't meet the standard	Modify the is wrong data again
E213	Single block is too long	The whole line number of part program is over the range	Modify the workpiece program
E214	Use command independently	some commands are in an alone block without others	Modify the source program
E215	Lack of line number	The specified program line number of command M96, M97, M98 or in the condition statement doesn't exist in this program i.e. the system has not found the line number for the call or the transfer	Modify the source program, input the correct block line number
E216	Line number repeat	Repeated program line number in program makes the block M96/M97/M98 or in the condition statement not be identified i.e. the program has more than two call or transfer needed line number	There is repeated program line number in modifying program
E217	Program stock data has err	System EMS memory error or program error	Need the professional to check the system all sided
E218	Errors in variable NO.	The variable number exceeds the set range	Using the usable variable number
E219	Edit area is unusable	System battery is shortage	Need to clear out program, Edit the program again
E220	Sentences calculate, divisor is 0	In the statement operation, the divisor is zero	Modify the divisor in macro command, not to be zero
E221	Not assign a value to variable	Input interface variable is read only, can't evaluate it i.e. input	Modify source program
E222	Program executes, information lost	The data has problem in program translate and edit process	Power on again
E224	Buffer area file too big	The editing single program is too big	Modify source program
E226	Coordinate fields repeat	Coordinate field repeat	Delete one field of them
E227	Data format wrong	Data format error (such as: F1r1, G1r, M1r, T2r), or commands like G1r, M1r, T2r are used.	Modify source program

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E228	Program hasn't end command	The program don't have the ending command M02 or M20 or M30	Modify source program, add the ending command
E233	No user-defined command	At first make program for the needed user-defined command M50, then harden it	Write M50 program, then harden it
E234	Data field repeat	The same field in block is input repeatedly	Delete the repeat data field
E240	Too many program nested calling layers	Excessive nested call in the program	Modify source program
E241	#254 not use traverse com	254 program has used motion command	Modify command or system parameter
E243	Too many data after decimal	The data behind the decimal point exceeds the limited range	At most three data is behind the decimal point
E244	Too many decimal in data	Wrong data is input i.e. 0.343.44	Modify data
E245	Lack of data after decimal	The decimal point is input, but no data followed	Delete the decimal point
E246	Fail to input negative in this field	The negative is used in this field	Check the notebook, modify the data
E247	Too much leading zero	The data like 000033 is used	Omit the needless zero in the front
E252	No replacing variable in M command	Commands such as M81 Ir1 are used.	Modify source command
E254	Forbidden M3/M4 com in M47	The M3/M4 command is used in M47 state	Using the command after change the state
E255	Undefined		
E257	No using the type of variable	In the special situation, this kind of variable is used in illegal	Modify source command
E258	Beveling data is too big	So large chamfer data can't be carried in this path	Reduce the chamfer data
E259	Editing is cut off	Reset key is pressed in program	
E260	No Y axis traverse in M48	When the Y axis has exchanged the spindle, the moving command can't be used in Y axis	Modify the program according to the real situation
E263	Use pointer variable wrongly	The pointer variable has special explanation, it must be used carefully	Modify program
E265	No calling in monitor macro	Call command is used in process monitor r7000 ~r7009 commands	Using jumping P or other method to control
E266	Pointer variable points invalid NO.	The pointer variable points to the forbidden variable	Check the program, then modify it
E268	Traverse is 0, retract too big (traverse amount is 0 or retraction is too big)	When making the G90 and G94 command program, the starting point and end point don't meet the regulation	Modify program
E272	S value can't be 0 in G50	Used the G50 command, it doesn't meet the program regulation	Modify the S value
E273	Macro value is negative	The variable value is negative, but the command need it be positive number	Modify program
E280	No relevant command in executed program	When executing G66/G67, G67 can be executed only after G66 is executed at least once; or G67 can be used when G66 is used at least once in programming,	Re-select program; or modify program
E282	Calling command of this type not allowed in No. 254 program	Custom command forbids motion command codes. Not conforming to the programming regulation	Modify program or parameter
E283	Floating point not allowed in operation of this type	The floating point cannot be involved in the logic operation. Not conforming to the programming regulation	Modify program

6.4.2 Alarm in Program Compound Check (i.e. E600~699)

Alarm in program check, means in the workpiece program the explanation of the command which is collided with system parameter. When the workpiece program is treated isolated, there is no problem. It needs to analyze the program from the whole auxiliary parameter and the setting of interface parameter, then to modify the program and parameter setting to eliminate the alarm.

List of alarm in program check

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E601	Illegal use Sxxxx：Sxxxx out of gear range	The Sxxxx command is over the parameter assured gear position range	Modify it to the gear position value in correct range or modify the P410 parameter
E602	Illegal use M21： No define M21 pin Para	The M21 interface parameter doesn't specify the exact chip pin	Set the interface parameter P500 is effective, or change the program command
E603	Illegal use M22： No define M22 pin Para	The M22 interface parameter doesn't specify the exact chip pin	Set the interface parameter P500 is effective, or change the program command
E604	Illegal use M23： No define M23 pin Para	The M23 interface parameter doesn't specify the exact chip pin	Set the interface parameter P501 is effective, or change the program command
E605	Illegal use M24： No define M24 pin Para	The M24 interface parameter doesn't specify the exact chip pin	Set the interface parameter P501 is effective, or change the program command
E606	Illegal use M91： No define M91 pin Para	The M91 interface parameter doesn't specify the exact chip pin	Set the interface parameter P517 is effective, or change the program command
E607	Illegal use M92： No define M92 pin Para	The M92 interface parameter doesn't specify the exact chip pin	Set the interface parameter P517 is effective, or change the program command
E608	Illegal use M93： No define M93 pin Para	The M93 interface parameter doesn't specify the exact chip pin	Set the interface parameter P518 is effective, or change the program command
E609	Illegal use M94： No define M94 pin Para	The M94 interface parameter doesn't specify the exact chip pin	Set the interface parameter P518 is effective, or change the program command
E610	Illegal use M78： Command invalid	The function of M78 is set to be invalid	Set the interface parameter P409_d4 is effective, or change the program command
E611	Illegal use M79： Command invalid	The function of M79 is set to be invalid	Set the interface parameter P409_d4 is effective, or change the program command
E612	Illegal use M41： Command invalid	The function of M41 is set to be invalid	Modify P410_d6=1(frequency conversion) or P410_d7=0(electrical level), or change the program command
E613	Illegal use M42： Command invalid	The function of M42 is set to be invalid	Modify P410_d6=1(frequency conversion) or P410_d7=0(electrical level), or change the program command
E614	Illegal use M43： Command invalid	The M43 function is set to be invalid	Modify P410_d6=1(frequency conversion) or P410_d7=0(electrical level), or change the program command
E615	Illegal use M44： Command invalid	The M44 function is set to be invalid	Modify P410_d6=1(frequency conversion) or P410_d7=0(electrical level), or change the program command
E616	Illegal use M32： Command invalid	The M32 function is not expanded to be effective	Set P506=1, or change the program command
E617	Illegal use M33： Command invalid	The M33 function is not expanded to be effective	Set P506=1, or change the program command

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E618	Illegal use M10： Command invalid	The M10 function is set to be invalid	Modify P409_d7 to be effective , or change the program command
E619	Illegal use M11： Command invalid	The M10 function is set to be invalid	Modify P409_d7 to be effective , or change the program command
E620	Illegal use r10xx： No program pin	The input chip pin by variable is occupied by other function, or the parameter forbids to use the input chip pin statement program	Modify P412_d1 to be effective, or change the variable corresponding input chip pin or change the program command
E621	Illegal use r20xx： No program pin	The input chip pin by variable is occupied by other function, or the parameter forbids to use the output chip pin statement program , or used the user-defined changing tool method	Modify P412_d1 to be effective, or change the variable corresponding input chip pin or change the program command, or modify parameter P318=9
E622	No traverse command in M61 program	The parameter setting: the M61 program forbids the motion command	Change the position parameter P404_d2 to be allowed, or change the program to don't include motion command
E623	Illegal use M47	The switch setting in spindle and Y axis is not	Modify P404_d4=1, or change the program command
E624	Illegal use M48	The switch setting in spindle and Y axis is not	Modify P404_d4=1, or change the program command
E625	Illegal use M04	The spindle reversion signal is set to be without	Modify P404_d2=1, or change the program command
E628	Tool type is 9, T command need single block	When the tool post type is 9, the T command doesn't have separate section	Modify program
E629	Illegal use Y command	The control axis is set to be Y axis	Modify P405_d1=1, or change the program command
E630	Cut NO. exceed	In the program the tool number is over the setting range	Modify P319, or change the program command
E631	Illegal use G96	When the spindle S control is set to be gear shifting mode, the constant line speed cutting	Modify P410_d6=1, or change the program command
E632	No define G31I input interface, invalid	Don't set the G31I interface parameter, or the setting is wrong	Set the P532 to be effective, or change the program command
E633	Illegally use the X command	The controllable axis is set to any one other than the X axis	Modify P405_d0=1, or alter workpiece programming
E634	Prohibit X axis G06 mode	Parameter P419_d2=0	Modify P419_d2=1, or alter workpiece programming
E635	Prohibit Y axis G06 mode	Parameter P419_d1=0	Modify P419_d1=, or alter workpiece programming
E636	Prohibit Z axis G06 mode	Parameter P419_d3=0	Modify P419_d3=1, or alter workpiece programming
E637	Chuck control disabled, prohibit call the M12	Parameter sets the prohibition control chuck	Modify P409_d7 =0 enabled, or alter workpiece programming
E638	Tailstock control disabled, prohibit call the M80	Parameter sets the prohibition control tailstock	Modify P409_d4 =0 enabled, or alter workpiece programming

6.5 Alarm Table in JOG OR AUTO Working Mode (i.e.E300~ E499)

The Alarm type in JOG and AUTO is divided to: Alarm in executing relative operation and The relative Alarm in executing statement two kinds.

6.5.1 Alarm in Executing Relative Operations (i.e E300~E399)

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E302	Z driver unit alarm	Z drive unit alarm	Check Z axis drive unit, resolve the failure to power-on again
E303	X driver unit alarm	X drive unit alarm	Check t X axis drive unit, resolve the failure to power-on again
E304	Y driver unit alarm	Y drive unit alarm	Check Y axis drive unit, resolve the failure to power-on again
E305	Positive hardware limit alarm	X or Z positive limit switch is closed	X or Z negatively moves in JOG working mode till the alarm is released
E306	Negative hardware limit alarm	X or Z negative limit switch is closed	X or Z positively moves in JOG working mode till the alarm is released
E307	Z machine (+) software limit alarm	Z machine coordinate(Zm) exceeds the positive software limit alarm value (P015)	X or Z positively moves in JOG working mode, or set the machine coordinate to zero again, release the alarm
E308	X machine (+) software limit alarm	X machine coordinate(Zm) exceeds the positive software limit alarm value\(P017)	X negatively moves in JOG working mode, or set the machine coordinate to zero again, release the alarm
E309	Y machine (+) software limit alarm	Y machine coordinate(Zm) exceeds the positive software limit alarm value(P019)	Y negatively moves in JOG working mode, or set the machine coordinate to zero again, release the alarm
E310	Z machine (-) software limit alarm	X machine coordinate(Zm) exceeds the negative software limit alarm value(P016)	Z positively moves in JOG working mode, or set the machine coordinate to zero again, release the alarm
E311	X machine (-) software limit alarm	X machine coordinate(Zm) exceeds the negative software limit alarm value(P018)	X positively moves in JOG working mode, or set the machine coordinate to zero again, release the alarm
E312	Y machine (-) software limit alarm	Y machine coordinate(Zm) exceeds the negative software limit alarm value(P020)	Y positively moves in JOG working mode, or set the machine coordinate/machine zero return again, release the alarm
E313	Z nose (+) software limit alarm	Z tool nose coordinate exceeds the positive limit alarm value(P009)	Z negatively moves in JOG working mode, or execute the tool setting/program reference point return again, release the alarm
E314	X nose (+)software limit alarm	X tool nose coordinate exceeds the positive limit alarm value(P011)	X axis negatively moves in JOG working mode, or execute the tool setting/program reference point return again, release the alarm
E315	Y nose (+) software limit alarm	Y tool nose coordinate exceeds the positive limit alarm value(P013)	Y negatively moves in JOG working mode, or execute the tool setting/program reference point return again, release the alarm
E316	Z nose (-)software limit alarm	Z tool nose coordinate exceeds the negative limit alarm value(P010)	Y positively moves in JOG working mode, or execute the tool setting/program reference point return again, release the alarm
E317	X nose (-)software limit alarm	X tool nose coordinate exceeds the negative limit alarm value(P012)	X positively moves in JOG working mode, or execute the tool setting/program reference point return again, release the alarm
E318	Y nose (-) software limit alarm	Y tool nose coordinate exceeds the negative limit alarm value(P014)	Y positively moves in JOG working mode, or execute the tool setting/program reference point return again, release the alarm
E319	Output frequency too high	Z/X/Y pulse output frequency exceeds the system specified range	Reduce the moving speed or modify the pulse multiplication ratio, division coefficient(P203, P204) to the proper value
E320	Thread spindle speed too high	the value of spindle speed multiplying the thread pitch(P) is more than max. speed limit of cutting feed (P113)	Reduce the feedrate or modify P113 value to the enough

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E321	Can't execute G32 in DRY	The system cannot execute G32 in DRY RUN working mode	Deleted G32, then execute the program in DRY RUN working mode
E322	G34: Screw pitch is out of range	Before G34 thread machining, the pitch is changing, and it exceeds the permitted range or be a negative value when the end point is reached.	Modify the machining program
E323	Coordinate flow alarm	The coordinate exceeds the top value of allowance data	Programming modification, avoid the coordinate freely moves toward one direction.
E335	Tool-post NO. alarm	After tool change completed, the tool selection number isn't the target number	Check the target tool selection signal is in normal or not
E336	No changing tool	The parameter set the tool changing forbidden	Modify the forbidden tool changing parameter P403_d7
E337	Detect tool post signal overtime	In tool post rotation (CCW) time in tool change, it can't inspect the tool selection number	Check the target tool selection signal is in normal or not, or delay the time for tool changing in properly
E338	Chuck doesn't clamp, no spindle	Forbid starting the spindle when the chuck is released in spindle and chuck locking each other	Execute the chuck clamping operation to clamp the workpiece
E339	Spindle doesn't stop, no chuck	When the spindle and chuck are locking each other, don't operate the spindle without stopped the spindle	Stop the spindle, then operate the chuck
E340	Forbid function of chuck	The parameter set chuck control forbidden	Modify parameter P409_d7
E341	Spindle doesn't stop, no tail	When the spindle and chuck are locking each other, don't operate the tailstock without stopped the spindle	Stop the spindle, the operate the tailstock
E342	Forbid function of tail stock	The parameter set tailstock control forbidden	Modify the parameter P409_d4
E343	M10 no respond	The system cannot check the valid chuck champing in position signal RM10 in controlling chuck champing	Check the chuck clamp selection signal is in normal or not
E344	M11 no respond	The system cannot check the valid chuck releasing selection signal RM11 in controlling chuck releasing	Check the chuck releasing selection signal is in normal or not
E345	M78 no respond	the system cannot check the valid tailstock going-forward in-position signal RM78 in controlling tailstock going-forward	Check the tailstock signal is in normal or not
E346	M79 no respond	The system cannot check the valid tailstock going-backward in-position signal alarm 79 in controlling tailstock going-backward	Check the tailstock going-backward signal is in normal or not
E347	The non-frequency spindle	The system can't execute the constant surface speed command G96 or gear shifting command M41~M44 in non-frequency spindle	Modify the spindle collocated parameter P410_d6
E348	The program is not ready	Program error, execution is forbidden	Modify the program, resolve error
E349	Use [machine zero return] key	When the machine is back to zero function, the modifying manual is forbidden	Using " MACHINE ZERO RETURN" key to zero
E350	S exceed limit	The Sxxx value exceeds the gear shifting range in the non-frequency spindle gear shifting	Modify the S value to the correct gear value
E351	Spindle is controlled by Y axis	When the spindle is controlled by Y axis, the start and stop of the spindle can't be controlled in JOG	Execute M48 to get back the spindle function
E352	Spindle Y, levels can't be changed	The spindle can't execute the gear shifting command when it is controlled by Y axis	Execute M48 to get back the spindle function
E353	M47/M48 function doesn't work	The M47/M48 function forbidden is set in parameter when there is no Y axis(P405_d1) or Y axis can't be changed (P410_d4)	Modify parameter P405_d1 and P410_d4

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E354	Undefined		
E355	254 program error	254 program error, the machining program is forbidden to execute	Modify 254 program, resolve error
E356	Input/output multiple use	The same input/output port is used for the different signal.	The repeated used signal at the same port is separately employed different ports.
E357	Feeding override is 0.	The override is zero in feedrate	Adjusted to feedrate override nonzero
E358	Spindle running is abnormal.	The spindle rotate speed is checked to be zero after the spindle is started	Check the spindle relative part, resolve the error or modify parameter **P402_d6** to forbid its function
E359	Alarm: pressure is low.	The system has checked the low pressure and lasted the parameter P332 specified time	Check the hydraulic pressure equipment, resolve the error
E360	Alarm: protecting door isn't closed	The safety door don't close in AUTO	Close the safety door
E361	Material convey alarm	The system has checked it's out of rod in M20, automatism stop alarm	Change the rod, then go on the machining
E362	Lubricating doesn't start	The system executes the M32/M33 command without lubricating function	Set the parameter P506 to be the corresponding lubricating control interface
E363	Tool NO. exceeds limit	The target tool number exceeds the parameter P319 specified value in changing tool	Modify tool number or modify P319 to be the maxima system l tool number
E364	Tool post release overtime	The tool pose champing signal is effective always in tool changing and releasing tool post	Check the tool pose champing signal and resolve error
E365	Y axis is speed control mode	The spindle and Y axis allow change(P410_d4), but the system don't allow to control the Y axis moving in current spindle non-position control mode	Execute M47 first, change the spindle to location control mode
E366	Clamp in-position signal abnormal	When the following chuck respond signal checking alarm function is open, the chuck champing in-position signal is invalidation	Control the chuck champing again or resolve the chuck error
E367	Release in-position signal abnormal	When the following chuck respond signal checking alarm function is open, the chuck champing in-position signal is invalidation	Control the chuck champing again or resolve the chuck error
E368	Tail stock FW signal abnormal	When the following tailstock respond signal checking alarm function is open, the tailstock going-forward in-position signal is invalidation	Control the tailstock going-forward again or resolve the chuck error
E369	Tail stock BK signal abnormal	When the following tailstock respond signal checking alarm function is open, the tailstock going-backward in-position signal is invalidation	Control the tailstock going-backward again or resolve the chuck error
E370	P319 (number of tools) and P320 (tool position signal number) are not equal	The parameter setting P319≠P320 in tool changing, the position parameter **P408_d7**=0	Set the parameter **P408_d7**=0-1 again or let P319 equal to P320
E371	External start signal abnormal	The outside starting key always isn't released when the system changes to JOG or AUTO working mode	Check the outside startup signal and resolve the error
E372	Chuck foot switch signal abnormal	The chuck foot switch always isn't released when the system changes to JOG or AUTO working mode	Check the chuck foot switch signal and resolve the error
E373	Tail foot switch signal abnormal	The tailstock foot switch always isn't released when the system changes to JOG or AUTO working mode	Check the tailstock foot switch signal and resolve the error

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E374	G31 monitor interface undefined	The parameter P532 is undefined(G31 input checking)	Set interface parameter P532 as a fixed pin
E375	Spindle doesn't stop	When the system is in the tapping in JOG working mode, the spindle doesn't stop or the rotate speed isn't zero	It's allowed to enter the tapping in JOG working mode when the spindle has stopped and the rotate speed is zero
E376	Manual tapping speed is too high	The manual tapping rotate speed is too fast	Reduce the tapping rotate speed in properly
E377	Commands have been undefined	Used the undefined command M50~M74	Modify the program
E378	External pause signal abnormal	The external pause key is not released during MANUAL/AUTO mode switching	Check the external pause signal
E379	Feed hold switch is not reset	Workpiece program cannot be executed when the feed hold switch is not cancelled	Check feed hold signal
E380	Spindle/feed hold switch is not reset	Workpiece program cannot be executed when the spindle/feed hold switch is not cancelled	Check spindle/feed hold signal
E381	Do not perform the M47/M48 if the Y axis is still executed	The M47/M48 is performed when Y axis is still in the movement state.	Modify the programming, correct the error.
E382	Do not open the spindle if the tailstock does not ready.	The spindle is started when the tailstock does not already.	Check the tailstock and then remove the fault
E383	Do not operate the spindle in M87 state	Try to start the spindle in the M87 orientation state (regardless whether the orientation is completed or not).	Firstly perform the M88, and then perform the spindle command after retreating from the M87 state.
E384	Disabled in M87/M88	Parameter P343 is set to 0, M87/M88 function is still closed.	Set the parameter P343, open the M87/M88 function.
E385	Z axis movement is restricted by external control.	Z axis movement is restricted by external control (It is specified by parameter P538).	Check whether the external connection signal is normal.
E386	X axis movement is restricted by external control	X axis movement is restricted by external control (It is specified by parameter P539).	Check whether the external connection signal is normal.
E387	Y axis movement is restricted by external control	Y axis movement is restricted by external control (It is specified by parameter P540).	Check whether the external connection signal is normal.
E388	M84 alarms when inspecting the I signal overtime	The overtime alarm of M84 inspection signal	Check whether the external connection signal is normal.
E389	Do not perform the M13/M14 if the spindle does not stop	Spindle in the M03/M04 state	Perform the M13/M14 after stopping the spindle.
E390	Fail to perform the M13/M14 during the movement of the Y axis	The spindle start and Y axis is interlocked with the spindle clamping when P410_d4=1. Forbid performing the M13/M14 command in the Y axis movement state.	Set the bit parameter P410_d4=0, or it only can be performed after the Y axis movement is completed.
E391	Forbid starting the spindle in M13 state	Start the spindle in the M13 state	Perform the M14 to retract the spindle clamping signal, and then start the spindle
E392	Fail to perform the M13/M14 because the spindle is performing now.	Spindle in the working state	Perform the M13/M14 after the spindle execution is completed.
E393	Forbid the Y axis movement command in G09 state	G09 Y0, fail to perform the G06 in monitoring G09 Y1, forbid to move Y axis	Modify the programming, correct the error

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E394	Forbid the Y axis movement command in M13 state	The spindle start and Y axis is interlocked with the spindle clamping when P410_d4=1. Try to perform the Y axis movement in M13 spindle clamping state.	Set the bit parameter P410_d4=0, or perform the M14 firstly, and then the Y axis movement command.
E395	Forbid the Y axis movement command in M47 state	System in the M47 H1 state	The system output API LEVEL signal is “0”.

6.5.2 Relative Alarm in Executing Statement (i.e.E400~ E499)

Alarm in statement program means in the program by statement, the alarm is made by the wrong statement command in executing the workpiece program, the alarm can be resolved by inputting the correct command.

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
E400	Monitor variable not initialize	The system doesn’t evaluate and set the judgment condition to the process monitor before startup	Modify workpiece program
E401	In macro, divisor is 0	The divisor is zero in statement operation	Modify the divisor in macro command, let it don’t be zero
E402	not support monitor calculation	The process monitor assignment statement has assignment operation which is not supported by the system, i.e. square root, sine operation	Modify program
E403	r5008 data error	Evaluate the variable r5008 illegal value	Modify program
E404	Variable data r6xxx error	Illegally used the undefined r6xxx series variable	Modify program
E405	Variable data r4xxx error	Illegally used the undefined r4xxx series variable	Modify program
E406	r4003 can’t be written	The illegal written operation is made to variable r4003(spindle real-time speed)	Modify program
E407	r4007 can’t be written	The illegal written operation is made to variable r4007(spindle real-time speed)	Modify program
E408	r4008 can’t be written	The illegal written operation is made to variable r4008(spindle real-time speed)	Modify program
E409	Not assign a value to r6xxx in monitor	The condition statement in process monitor evaluates the r6xxx series variable	Modify program
E410	r711x Invalid data	Write the undefined value to r711x series variable	Modify program
E411	r4002 can’t be written	Written operation is made to the read only machining time variable r4002	Modify program
E412	The variable doesn’t support this kind of operation	The system non-supported operator is used in the assignment statement	Modify program
E413	The unknown process monitor description	The system non-supported judgment condition is used in the statement of process monitor description	Modify program
E414	Unknown pointer variable	The pointer variable points to the non-common variable or the exceeded range pointer variable is used	Modify program
E415	The variable r4004 can’t be written	Making written operation to reading variable r4004	Modify program
E416	The variable r4012 can’t be written	Making written operation to reading variable r4012	Modify program
E417	Fail to write the variabler4013	Write the READ variable r4013	Modify the workpiece programming, correct the error
E418	Fail to write the variabler4014	Write the READ variable r4014	Modify the workpiece programming, correct the error
E419	Fail to write the	Write the READ variable r4015	Modify the workpiece

Alarm No.	Alarm prompt	Alarm reason	Troubleshooting
	variabler4015		programming, correct the error
E420	Fail to write the variabler4016	Write the READ variable r4016	Modify the workpiece programming, correct the error
E421	Try to modify the read variable	Try to modify the READ macro variable	Modify the workpiece programming, correct the error
E422	Alarm end monitoring	Alarm ends when monitoring command is performed.	Releasing alarm

CHAPTER SEVEN STATEMENT PROGRAMMING

This system has provided the program method similar to the advance language; it can realize the variable assignment, arithmetic operation, logic judgment and conditional transfer. Using the statement and variable program can come true the function which can't be made by the common G and M command.

7.1 Variable

7.1.1 Variable Expression Method

The variable value can be set by the program command assignment or by key directly. Multi-variable can be used in a program and they can be distinguished by variable number.

The variable expression method

Using small letter "r" + variable number (four digit integer) to express, the precursor zero can be omitted. Such as:

r5: named number 5 variable; r1003: named number 1003 variable

7.1.2 Classification of Variable

The variable in this system can be divided by function and purpose as: common variable, pointer variable, input/output interface variable, special variable in system inner, register variable of key scan command, register variable of showing command, register variable of command, process monitor/ process monitor management variable, pulse signal/pulse signal monitor management variable etc.

Different variable has different function and purpose, its explanation and value range are different too. Following is the instruction in classify.

Variable type	Variable range	Remark
Common variable	r001～r299	
pointer variable	r9000～r9049	
input interface variable	r1001～r1032	
output interface variable	r2001～r2032	
register variable	r5001～r5004, r5008	
system variable	r4001～r4023, r6001～r6006, r6101～r6164, r6201～r6264	
process monitor variable	r7000～r7009	
process monitor management variable	r7010～r7019	
pulse signal monitor variable	r7100～r7103	
pulse signal monitor management variable	r7110～r7113	
variable transfer register variable	r7900	

7.1.2.1 Common Variable

The variable number range of common variable is from 001 to 299, the total is 299 pieces.

The common variable is used to store a data which can participate in operation and also be quoted by data field of G command etc.

The data value range of command variable: -999999999～999999999.

◆ **common variable reference**

In the workpiece program, the system uses the variable to change the data field value. (Only the common variable can be used like this.)

Format: <address> + “ri”, means to treat the variable value as field value

For example: Fr003: when r003 equals 15000, its function is same to F15 command

Zr010: when r010 equals 2500, its function is same to Z-2.5 command

K r 010 ; when r010 equals 2500, its function is same to K2.5 command

S r 003 ; when r003 equals 2000000, its function is same to S2000 command

【For example】

```
N0010 r001=-3700
N0020 r002=150000
N0030 G00 Z100 X80
N0040 G01 Ur001 Fr002   ; same to U-3.700   F150.000 command function
N0050 G01 Wr001
N0060 M02
```

【Explanation】

1) The variable value doesn’t include decimal point, the variable value is get from representative value riding 1000, when it is replaced to the coordinate etc field with decimal point, the variable value divides 1000, then the coordinate date including three decimal is get;

2) In the command symbol G, M, T the variable can’t be quoted; Gr003 can’t be used for program;

3) The field of program line number (i.e. P, Q etc) can’t quote variable;

4) The field L showing times and field H showing location sign can’t quote the variable;

5) Only the common variable can be quoted by command field, otherwise alarm;

6) When the program is in AUTO working mode, when the variable is used in main program and also the transferred subprogram(i.e.: M50~M74 command), please pay attention that the variable in the intercross using and changing of main program and subprogram will impact the result of program running.

◆The characteristic of common variable：

When the system executes the program in AUTO working mode, the variable value may change along with the program path. The common variable has following characteristics:

1) The system makes different initiation disposal according to the different variable section number.

Among them: r001～r040：The variable parameter table offers the initial value for it (set by the user according to the requirement)

r041～r099：The INITIAL VALUE of this group is 0 in AUTO mode when the program is started.

r100～r199：Never attempt to clear except the boot-strap.

2) In the AUTO working mode, before executing the first program and pressing the CYCLE START key, the system will put the parameter table P600 ~P639 into variable r001~r040 as the initial value in AUTO; the variable r041~r099 is cleared.

3) After executed M20, in the recycle, r001～r040 are evaluated the initial value by the P600～P639 in parameter table again; r041～r099 and r100～r199 are not cleared.

4) The initial value of r100～r199 is zero after power-on, later it won't be cleared by itself. But it can change their value in JOG or AUTO working mode immediately. Their initial value can be set by the following method (using this variable in program, when the variable initial value isn't evaluated before using, the program won't have the confirmation state, suggest the user carefully). When the user is using these variable and needs the initial value, the initial value of r100~r199 can be set in JOG working mode. In the JOG or AUTO working mode and non running the program, press ⇧ will call the macro variable display window and express all the macro variable in the program, press ⇨ and ⇩, ⇧ will let the pointer pointing the needed modifying variable, press ENTER key to choose the variable and input the data, then press ENTER key to confirm again. The variable value is forbidden to modify in the program running process.

◆Double-precision floating-point variable__common variable r200~r299

Double-precision floating-point variable belongs to common variable; its range is 200~299, 100 in total.

As data flow may easily occur during common variable (r001~r199) calculation, the use of double-precision floating-point variable can solve this problem. Range of double-precision floating point: 64-digit with sign

【Explanations】

1) Do not use the double-precision floating point variable in the monitoring;

2) Double-precision floating-point variable cannot be in code field, otherwise, an alarm will occur.

3) Refer to instruction of common variable (r001~r199).

【Characteristics】

In AUTO mode, double-precision floating-point variable may vary with program path. It has following characteristics:

1) In AUTO mode, in the INITIAL STATE of execution, the variables are 0.

2) The variable value will not clear to 0 when recycle after executing M20.

3) In MANUAL or AUTO mode, press ⇧ to call macro variable; the window does not display double-precision floating-point variables.

7.1.2.2 Pointer Variable

The variable number range of pointer variable is from 9000 to 9049, total 50 pieces.

The pointer variable is specially to point the address of a common variable. To evaluate it means to evaluate all it pointed common variable; and using its value means to use all it pointed common variable's value. The pointer variable can participate the operation or reading judgment.

◆ **The method of creating and changing pointer**

The pointer must be created correctly and points to a certain given common variable before using the pointer variable. The pointing sign are "-" and ">" and a blank is allowed between them.

The variable format of creating and changing pointer:

Pointer variable name -> expression

Such as: r 9001 -> 1 ; expression of the r9001 pointer variable pointing to r001 common variable;

r 9002 -> 199 ; expression of the r9002 pointer variable pointing to r199 common variable;

r 9003 -> r100 ; when r100 equals to five, it's the expression of r9003 pointing to r005 common variable

r 9003 -> r 9003+1 ; expression of r9003 pointing to next common variable, when it points r5 at first, then now it points to r6;

r 9003 -> r 9003-1 ; expression of r9003 pointing to previous common variable, when it points r6 at first, then now it points to r5;

◆**The characteristic of pointer variable:**

1) At first set up the pointer variable and let it point to a certain specified common variable, then make the read/ written operation to it or use for judgment, otherwise alarm;

2) In setting up the pointer variable, the value range of is from 1 to 199 (It is to say it can point r1 to r199), when it's exceeded, it will alarm;

3) The statement about setting up or changing the pointer variable, only can be written in a separate section, otherwise alarm; the expression in the right side of pointing symbol can be the addition and subtraction operation between two of them;

4) The pointer variable only can be used in the numerical value operation, can't be quoted by the data field like G etc command; such as Ur9000, alarm;

5) In the statement of assignment, conditional judge, to evaluate the pointer variable expresses to evaluate the common variable which is pointed by it; however reading the pointer variable expressed to read the value of the common variable which is pointed by it.

◆**The use of pointer variable：**

【Program demonstration】

In the following program, Z/X axis will store the coordinate of the position to the common variable in moving each position, it totally has stored 10 set coordinate; then take out them one by one and go back to the initial position by the original path. The following is come true by the pointer variable.

```
N0000   G00 Z200 X200          ;
N0010   r9000 -> 1             ; expression of the r9001 pointer variable pointing to r001
```

```
                                    common variable;
N0020   M98 L10 P1000          ;
N0030   M98 L10 P2000          ;
N0040   M02                    ;
N1000   r9000= r6004           ; In the first calling, store the Z coordinate in this position to r1
N1010   r9000 -> r9000+1       ; In the first calling, the pointer +1 points to r2
N1020   r9000= r6005           ; In the first calling, store the Z coordinate in this position to r2
N1030   r9000 -> r9000+1       ; In the first calling, the pointer +1 points to r3
N1040   G00 W-25               ;
N1050   G00 U-15               ;
N1060   M99
N2000   r9000 -> r9000-1       ;In the first calling, the pointer -1 points to r20
N2010   r190= r9000            ; In the first calling, take out the reciprocal second X coordinate
                                  from r20 and store it to r190
N2020   r9000 -> r9000-1       ; In the first calling, the pointer +1 points to r19
N2030   r191= r9000            ;In the first calling, take out the reciprocal second Z coordinate
                                  from r20 and store it to r191
N2040   G01 Xr190 F1000        ;
N2050   G01 Zr191              ;
N2060   M99
```

7.1.2.3 Interface Variable

This variable is correspond to input and output pin, specially is used to check and control the input/output pin signal of the system. The interface variable is divided into input interface variable and output interface variable.

◆**Input interface variable** (Read)

The system has 32 input interface variable, each variable is correspond to a input signal pin (the variable number r1001 ~r1032 is correspond to the signal pin number **UI01**～**UI32** in ranking)

Attention: the variable number r1000 is the integer state of 32 input interface; corresponding signal pin**UI01**～**UI32** from low to high.

The second state value of interface variable is 0 (low level) or 1(high level). The low level means the effective outside signal, express the pin contacted with 0V; the high level means ineffective.

The input interface variable is a set of read variable, can't be evaluated (input), otherwise alarm.

Through the program, read the value of input interface signal r1001~r1032 can know the pin corresponding signal level, and then it can be a judgment condition for the system to make the program skip.

【**For example:**】

```
r8=r1002      ;   read UI02 pin level; r8=0   or 1.
if (r1009=0) then(P1600)   ;   if the UI09 pin is read to be low level, then go to P1600.
r1007= r8     ;   system alarm: " Assignment cannot be done to read-only variables".
```

◆Output interface variable (read/write)

The system has 32 output interface variable, each variable is correspond to a output signal pin (the

variable number r2001 ~r2032 is correspond to the signal pin number **UO01～UO32** in ranking). The assignment on output interface variable can change the corresponding pin output state.:

When the system outputs “0”, the outside can form the conductive loop;

When the system output “1”, in the high resistance, the outside can’t form the conductive loop.

Read the output interface variable, the current assignment state of the output interface can be get.

【For example】

R2002=0 ; expression of outputting “0” to UO02 pin, the outside can form the conductive loop.

R2016=1 ; expression of output “1” to UO16 pin, in the high resistance, the outside can’t form the conductive loop .

r2007= r8 ; confirm the UO07 output state by the content of r8(zero or non zero)

If (r2016=1) then (P0100) ; if the output of current UO16 is read to be “1”, it will go to P0100.

【Explanation】

1) Only the output pin which is released by the standard function definition allows to the variable assignment output because of the system limitation, otherwise alarm; it only can make the statement program to pin with “UO” in the diagnosis interface, when it makes program to the defined input signal pin, it alarms.

2) In the system boot-strap and power-on beginning, the system will set r2001～r2032 to be “1”, the outside can’t form the conductive loop.

7.1.2.4 Keyboard Scan Register r5001

Keyboard scan register r5001 value range: 32 digits without sign

The different numerical value which is written by register has different meanings; the system executes the relative command according to the written numerical value. Please refer the below table:

The gathered table of r5001 executing keyboard scan commands: (the system forbids to use this variable in the process monitor description sentence)

Command No.	Meaning
1	keyboard scan once; When the key is pressed, the r5001 equals to the input value ; when there is no key pressed, r5001 equals to zero;
2	waiting for the keyboard input a character ; waiting until the key is pressed, and record it to r5001;
3	waiting for the keyboard to input a character, with cursor and character display; the key isn’t released, the cursor clue; the key is released, the value is recorded to r5001;
4	Waiting for the keyboard input a character and releasing press key;
5	Waiting for the keyboard input a character and releasing press key, with cursor and character display;
6	Waiting for the keyboard input a numerical value, and finished by ENTER; giving up by press ESC, then the input value will be zero.
Note: When the assignment is outside 1~6, it will be ineffective.	

【Explanation】

R5001 assignment is the working mode for setting keyboard register, the input value which is get

after executed the command is stored in register r5001, it can be read or use for the conditional judge .

【For example】

r5001=4; the system is waiting for the keyboard input a character, it will be executed until the key is released

If (r5001=51) then P1500; if press “3” , it turn to P1500 (the ASCII value of “3” is 51)

【Explanation】

1) In waiting for pressing, when the ESC is pressed, it will cancel the press key input, and executes the next statement.

2) The acceptable key symbol is as the below table, the other key input is ineffective. Notes: The input value is the corresponding ASCII value of this key symbol.

The r5001 acceptable key symbol and the input value corresponding table:

Input symbol	0	1	2	3	4	5	6	7	8	9	.	-	ENTER	ESC
input value	48	49	50	51	52	53	54	55	56	57	45	46	13	01
Input symbol	G	M	X	Z	S	T	U	W	F	I	K	D	R	
Input value	71	77	88	90	83	84	85	87	70	73	75	68	82	

7.1.2.5 Display Window Register r5002

Display window register: r5002 (the system forbids to use this variable in the process monitor description statement)

Through the operation to the display window register, the system can construct a window in the display screen and used to prompt the program execution step, alarm etc information.

【Explanation】

1) Set the size and color of the window before opening the display window; otherwise it’s the system windows default;

2) When the window is opened, when the window size etc parameter is reinstalled, the window content will be cleared automatism; the new setting is effective;

3) The reference of grounding and font color refers to the attached table in this chapter: the corresponding table of usually used color and command value;

4) In the AUTO running single block mode, when there is display character window currently, the key ESC is pressed, the window will be closed, and the program will exit the running. The different value is written in the register has different meanings; the system executes the relative command according to the written value. Please refer the following table:

The r5002 execution command table:

Command number	meaning(the user information only can be displayed in the display window
0	close the display window;
1	Open an empty display window;

2	when the window is not opened, call the display window which has opened last time;
10	Set the cursor position, the cursor points to the first line home;
11	Set the cursor position, the cursor points to the next line home;
12	Erasure the content from the current cursor position to the line end; the cursor position don't change;
13	Erasure all the content in window, the cursor points to the first line home;
32～126	Sending an ASCII character to the display window; (see ASCII command table)
201～209	The cursor is positioned to the specified line, the range is from 1 to 9;
210～230	The cursor is positioned to the specified line, the range is from 10 to 30;
1000～1099	send a set of character string to the display window; the mantissa 00~99 are character string number, it at most can display 100 sets character string;
2000～2999	Send a alarm number to the display window: display "alarm XXX", the range of XXX is from 0 to 999;
110001～110009	Set the line number of the display window (line 1 to 9); (the system windows default is 6)
110010～110040	Set the displayed character number in each line (10 to 40); (the system windows default is 30)
120000～120255	Set the window grounding or character grounding, 0 to 255 can be chose; (the system defaults white, the value is 255)
130000～130255	Set the character color, 0 to 255 are available (the system defaults black, the value is 0)
140000	Set the size of the character: 16*16
140001	Set the size of the character:: 8*16 (system widows default)
Note: when the assignment exceed the range, it is ineffective.	

【Use】

R5002 assignment is to set the display window register command; the data is uncertainly when read the register, it can't be used for the conditional judge .

【Example】

```
r5002=1        ;  open an empty display window
r5002=49       ;  send the character "1" to display window
r3=5           ;
r5002=49+r3    ;  send the character 6 which has deflected 5 position with "1" to the display window
r5002=2001     ;  the display window displays " alarm 001"
```

◆Expression of character string:

In the block, one of the below two expressions can be used to input the character string:

1) Common expression

Format: serial number+ blank + "S" + "display character string" ;

Example: when the character string 00 want to display "1~9 please press the key to choose the need repeated working procedure:", the block content is as following:

N5000 00 S 1～9 please press the key to choose the needed repeated working procedure

N5010 r5002=1000 ; display the set 0 character string r5010

2) ASCII code decimal expression

Format: serial number + blank + "D" + character code =blank +character code +blank + character code…

Example: when the 04 character string displays "1-9 GongXu：", the block content is as following:

```
N5000  04 D 49 95 57 32 71 111 110 103 88 117 58
N5010  r5002=1004 ;   display the fourth character string
```

【Explanation】

1) the serial number must be two digit figure, the range is from 00 to 99, total 100 set character string, and the lowered two digit command must correspond with the serial number, then it can display the content of this character string;

2) Behind the S blank, the system treats the content as character string; it can't add the note behind the character string;

3) Each set of character string won't exceed 40 characters (20 Chinese characters)

4) The character string can be written before or behind the program; it generally is written in the end and won't impact the program execution speed.

【Program Example】

It mainly demonstrates the explanation of keyboard scan register, display window register in the following program. Suppose there are three working procedure which each of them is moved to different position from start, then goes back to start. But the three working procedures are not executed by turn, they are chosen by press key temporarily according to the pop-up window prompt.

```
%101
N0000 G00 Z0 X0
N0010 G00 Z100 X100
N0020 r5002 = 110003              ;set the line number of the display window
N0030 r5002 = 110016              ;set the display character number of each line
N0200 r5002 = 1                   ;open an empty display window
N0210 r5002 = 140001              ;8*16   set the character size: 8*16
N0220 r5002 = 130000              ;set the character color: black
N0230 r5002 = 1004                ; set the 004 character string
N0240 r5002 = 11                  ;set the cursor position, the cursor points to the home of next
                                    line
N0250 r5002 = 140000              ;16*16 set the character size: 16*16
N0260 r5002 = 1006                ;display the 006 character string
N0270 r5002 = 11                  ; change line
N0280 r5002 = 130006              ; change color: blue
N0290 r5002 = 1005                ;display the 005 character string
N0300 r5002 = 130168              ; change the color: red
N0500 r5001 = 4                   ; Waiting for the keyboard input a character, release the key
                                    and then execute the next block
N0510 if(r5001 = 49) then(P1000)  ;If the character "1" is input, executes P1000
N0520 if(r5001 = 50) then(P2000)  ;If the character "2" is input, executes P1000
N0530 if(r5001 = 51) then(P3000   ;If the character "3" is input, executes P1000
N0540 if(r5001 = 48) then(P600    ;If the character "0" is input, executes P1000, ending
N0550 M97 P500
N0600 M02
N1000 r5002 = r5001               ; display the keyboard input character "1"
N1020 G01 Z50 X10 F2000
N1030 G00 Z100 X100
```

```
N1040 M97 P200
N2000 r5002 = r5001              ; display the keyboard input character “2”
N2020 G01 Z120 X50 F2000
N2030 G00 Z100 X100
N2040 M97 P200
N3000 r5002 = r5001              ; display the keyboard input character “3”
N3010 G01 Z160 X90 F2000
N3020 G00 Z100 X100
N3040 M97 P200
N4000  ;the note can’t be added behind the character string
N4000  ;the character string can be written finally, it won’t impact the execution speed
N5000 04 D 49 95 51 32 71 111 110 103 88 117 58
N5010 05 SINPUT
N5030 06 S       total three working procedure
```

7.1.2.6 Display Value Register r5003

Display value register: r5003 (in the description of process monitor statement, this variable is forbidden to use)

【Explanation】

The r5003 assignment is a command to display this value (it can be seen when the display window is open), the data is uncertainty when read the register, it can’t be used for the conditional judge .

【Example】

```
r5003=r032        ;  display the value in r032
r5003=r03+r01     ;  display the value of r03 adding r01
```

7.1.2.7 Graph Update Register r5004

Graph update register: r5004 (in the description of process monitor statement, this variable is forbidden to use)

【Explanation】

The r5004 assignment is a command to clear the process graph display area (it can be seen when the graph display window is open). Write the random number to r5004, clear the content of the system process graph display area.

【Example】

Graph update register: r5004 (it is forbidden to use this variable in the process monitor description statement)

Note: In the cycle process, the last process graph isn’t cleared; in order to let the user observe the new process path and graph in conveniently, in the ending of the process program, execute the command, the next process path and graph will be displayed again.

7.1.2.8 Program Control Register r5008

Program control register: r5008 (usually used for program description during monitoring, do not used in common programs). If data is uncertain when reading this register, it cannot be used.

Different values written in the register have different meanings and correspond to different functional commands.

Available commands in r5008

No.	Meaning
1	Pause: movement stops (equals to key CYCLE PAUSE); press CYCLE START to resume;
2	Switch between SINGLE/CONTINOUS mode (equals to key SINGLE); press CYCLE START to resume; (Usually used for program description during monitoring. If it is used in common program, M00 command is advised.)
3	Stop after cycle (movement stop after M20); press CYCLE START to resume;
4	Cycle starts (equals to key CYCLE START); usually used for description of process monitoring;
5	Set to SINGLE mode; press CYCLE START to resume;
6	Set to CONTINUOUS mode
11	Foot switch is allowed for chuck operation afterwards; M10/M11 switching; If spindle and chuck are interlocked, it is valid only under M05 command;
12	Foot switch is allowed for chuck operation afterwards; M10/M11 switching; If spindle and chuck are interlocked, it is valid only under M05 command and the actual speed is 0;
13	Foot switch is not allowed for chuck operation afterwards (except for SINGLE PAUSE); if the chuck operation is not finished yet, the next program cannot be executed;
14	Foot switch is allowed for chuck operation afterwards; M78/M79 switching; If spindle and chuck are interlocked, it is valid only under M05 command;
15	Foot switch is allowed for chuck operation afterwards; M78/M79 switching; If spindle and chuck are interlocked, it is valid only under M05 command and the actual speed is 0;
16	Foot switch is not allowed for chuck operation afterwards (except for SINGLE PAUSE); if the chuck operation is not finished yet, the next program cannot be executed;
17	Goes to next program if M10 is not finished; does not take up time;
18	Goes to next program if M11 is not finished; does not take up time;
19	Goes to next program if M12 is not finished; does not take up time;
20	Goes to next program if M78 is not finished; does not take up time;
21	Goes to next program if M79 is not finished; does not take up time;
22	Goes to next program if M80 is not finished; does not take up time;
23	Goes to next program if M3 is not finished; does not take up time;
24	Goes to next program if M4 is not finished; does not take up time;
25	Goes to next program if M5 is not finished; does not take up time;
40	Does not go to next program till all M commands are finished;

【Instructions】

1) During program execution, the operation of chuck and tailstock controlled by foot switch equals to inserting a work during the machining process. It does not take up time. If command in the same group is executed, it is invalid; the operation can be done only after the execution.

2) In statement or all-monitoring state, the operation of chuck and tailstock controlled by r5008 does not take up time;

3) Sometimes, the operation of chuck and tailstock controlled by r5008 will be invalid; for instance, when M11 is being executed, the operation is invalid;

4) If spindle and chuck or tailstock is not interlock, the operation of chuck and tailstock controlled by r5008 is valid regardless the spindle state and spindle speed.

5) In program end, reset, emergency stop state, the current command is cancelled automatically.

【Usage】

Assignment to r5008 means to choose program pause and start mode; usually it is used together with monitoring description; see the example;

【Example】

r5008=1 ; Program pauses immediately; press CYCLE START to resume;
r5008=3 ; Pause after cycle ends; press CYCLE START to resume;

7.1.2.9 System Special Variable Set 1

System special variable set 1: read/ write (in the process monitor description statement, read only, write forbidden).

Variable number	meaning
r6001	Z machine coordinate
r6002	X machine coordinate
r6003	Y machine coordinate
r6004	Z tool nose coordinate
r6005	X tool nose coordinate
r6006	Y tool nose coordinate
r6101…6164	Z tool offset 01…64
r6201…6264	X tool offset 01…64

【Explanation】

For the written variable, the assignment is a command to modify the parameter, in the same time it has kept the assignment and can be used for the read or conditional judge . In the common situation, don't use the system inner special variable. The system inner special variable must be used carefully.

【Example】

r6001=150000 ; modify the current Z axis machine coordinate to 150.000
If (r6005>3000) then P1500; if the current X tool nose coordinate is over 3.000, then turn to P1500
r6201= r6201+20 ; add 0.020 to the X tool compensation of the first set tool offset number
The system inner special variable value range: 32 digits with symbol

7.1.2.10 System special variable set 2

System special variable set 2 (used for the monitor description)

[Explanations]

1) The workpiece counter is Read/Write operation.

2) If the r4009 instant interference function is used in the programming, the "G", "M" character or blank displayed upper the left corner of the screen is indicated which programming branch transformation function is; "G" means r4009=71, "M" means r4009=77, blank means r4009=0. Here, the "G" or "M" is out of the relationship with the G and M commands, which is only the button mark of the instant interference function. The assignment of 71 or 77 of the r4009 equals to the button input "G" or "M", same as the branch transformation.

3) r4010 and r4011 cannot be assigned as negative, which is meaningless if they are set to 0. Its Max. assignment range is: 0~9999999 (that is, 9999.999s), and it can be regarded as read or conditional judgment. For example: r1=r4010 or if (r4010>1000) then P0020

4) Generally, the r4005 ~ r4008 is treated as tool-change judgment. For example: it is used in

the M60 self-defined tool-change programming (Parameter P318=9).

Variable number	Meaning	Property
r4001	Workpiece counter: the displayed workpiece on screen In the program execution, the counter will add 1 in auto when it meet a M02 or M30 or M20.	Read/ write
r4002	Time counter: workpiece processing time; unit: ms	Read
r4003	Spindle rotate speed recorder : the system checked current rotate speed; unit: r/min	Read
r4004	Spindle program rotate speed recorder : in execute the S command, the system will input the variable for automatism program rotate speed at first;	read
r4005	Target tool position number recorder: the system will input the tool position number to this variable automatically in executing T command;	Read/ write
r4006	The target tool offset number recorder: the system will input the tool offset number to this variable automatically in executing T command; 0≤tool offset number≤64	Read/write
r4007	The current tool position number recorder: after the tool changing finished every time, the system will input the tool position number to this variable automatically.	Read
r4008	The current tool offset number recorder: after the tool changing finished every time, the system will input the tool position number to this variable automatically. 0≤tool offset number≤64;	Read
r4009	instantaneous interference recorder: In the AUTO working mode, when the system has checked the recently press key “G” or “M”, it will input the corresponding ASCII value 71/77 to r4009; and display the character on the top left corner of the screen. Write random number except the 71/77, or press X key, or enter the AUTO working mode, the variable value will be cleared to zero. Using r4009, it can increase the function of two program branch transfer, used in the temporally specified man interference of machining process.	Read/ write
r4010	timer ： In the AUTO working mode, when the variable value isn’t zero, each ms will reduce one automatically, until the zero; it can be the time for conditional judge ;	Read/ write
r4011	timer： In the AUTO working mode, when the variable value isn’t zero, each ms will reduce one automatically, until the zero;	Read/write
r4012	spindle coder ： Read the spindle coder and it’s value range: among 0~ (four times of real coder LINE); when the coder LINE (parameter P209) is set to be 1200, the value range will be 0~4800.	Read
r4013	The spindle encoder position in manual tapping retraction.	Read
r4014	The spindle encoder linear number in the completion of screw thread acceleration	Read
r4015	The screw thread axis acceleration distance in the thread machining	Read
r4016	The offset angle in the screw thread machining.	Read
r4017	Record the current state of Z axis, the read resolution range: 1~4.	Read
r4018	Record the current state of X axis, the read resolution range: 1~4.	Read
r4019	Record the current state of Y axis, the read resolution range: 1~4.	Read
r4020	Record the spindle execution state, the read resolution range: 0~3.	Read
r4021	Record the chuck execution state, the read resolution range: 0~3.	Read
r4022	Record the tailstock execution state, the read resolution range: 0~3.	Read
r4023	The storage position from the M86 data receiving.	Read

【Use】

For the written variable, the assignment is a command to modify the parameter, in the same time the variable has kept its assignment, and it can be read or condition judgment.

【Example】

if (r4008=1) then P0060 ; If the current is No. 1 tool offset, it turns to P0060
if (r4001=1000) then P1500 ; If the process number equals to 1000, it turns to P1500
if (r4009=71) then P0050 ; If the current special transfer function key is “G”, it turns P0050 to execute

◆ **The application of the variable r4017~ r4019**

The variable r4017~r4019 are separately recorded the G06 current state of the Z, X, Y axis. This variable can be used for the Read, Conditional judgment or Monitoring description.

For example: r4017 records the current state of Z axis

r4017=1, Z axis G6 is moving;
=2, Z axis G6 stops after reaching the programming position, and the position confirms;
=3, Z axis G6 stops after inspecting the signal, and the position does not confirm.
=4, Z axis G6 abnormally stops, for example, G7, ESP etc. and the position does not confirm.

[For example]

```
N0000 G00 X100 Z100
N0010 G06 Y120 F1000
N0020 G04 D5
N0030 if (r4019=2) then(P0060) ; r4019 records the current state of Y axis. If the Y axis G6 stops after reaching the programming position, and the position confirms, turn to P0060.
N0040 G01 X30 F200
N0050 M30
N0060 G01 Z50 X51 F200
N0070 G0681 Y200 R10 D2 F500 L2
N0080 G01 X30 F200
N0090 M30
```

◆ **The application of the variable r4020~ r4022**

Variables r4020~ r4022 are recorded the execution state of the M03, M10, M78 commands etc.: the execution of each state, the completion of execution, unsuccessful of execution (or end in the middle way), user decides whether close part output based upon these state programming.

r4020 indicates M03/M04/M05 state
r4021 indicates M10/M11 state
r4022 indicates M78/M79 state

r4020~r4022= 0 ---Leisure/the completion of execution；
= 1 ---Performing…；
= 2 ---Execution error, waiting for trying again；
= 3 ---Execution error, command end (leisure).

[Example]

```
N0000 r7000=r4021 ; The state of 0# monitoring inspection variable r4021, which consists of the monitoring description with the following judgment statements.
N0010 if(r7000=0) then P1000 ； If the “chuck clamping executed (r4021=0)” is monitored, turn to P1000 programming operation
```

```
N0020 G00 Z300 X200   ；Firstly positioning to the initial point
N0030 M10             ；Chuck clamping
N0040 r7010=1         ；Open the 0# monitoring based upon the method 1, if the content meets,
                        immediately stop the movement and then turn to P1000.
N0050 G00 W-100 F1000；The tool closes to the workpiece from the longer distance;
N0060 r7010=0   ；If the last command is ended, but still not inspect, and then close the 0#
                  monitoring
N0070 G00 Z300 X200   ；Return to the initial point
N0080 M02
N1000 G50 Z200 X180    ；Confirm this point to machining start, reset the workpiece coordinate
                        system
N1010 M03 S1000       ；Spindle negative
N1020 G01 Z90 F500
N1030 G01 X0
N1040 G51             ；Recovery the workpiece coordinate system
N1050 G00 Z300 X200   ；Return to the initial point
N1060 M02
```

7.2 Statement

This system has assignment operation statement and conditional judge statement; they will be introduced as following.

7.2.1 Assignment Statement

This system offered assignment statement can come true the operation between two variables or value, the detail expression and operation are as below table:

Gathered table of assignment operation statement 7-1:

Command format	Function	Definition
rN=rA	assignment	give the rA assignment to rN variable
rN=rA + rB	Decimal addition operation	
rN=rA - rB	Decimal subtraction operation	
rN=rA * rB	Decimal multiplication operation	
rN=rA / rB	Decimal division operation	
rN=rA or rB	Binary or operation	
rN=rA and rB	Binary and operation	
rN=rA xor rB	Binary xor operation	
rN=sqr (rA)	Decimal square root	rN = $\sqrt{rA}$ (rA don't support negative)
rN=abs(rA)	Decimal absolute value	rN= | rA |
rN=rA % rB	Decimal remainder	rN = the remainder of (rA ÷rB)
rN = rA*rB/rC	Decimal multiplying and dividing operation	
rN=sqrf (rA rB)	Compound square root	rN = $\sqrt{rA^2 + rB^2}$
rN=rA*sin(rB)	sine	
rN=rA*cos(rB)	cosine	
rN=rA*tan(rB)	tangent	
rN=rA*atan(rB/ rC)	arc tangent	

In the above table: rN express the variable name of storing operation result; rA, rB, rC express the variable for running the operation, they can be constant too.

【Explanations】

1) The value, variable value, operation result don't have decimal point, the unit is 0.001;
For example: the assignment of Angle 45° must be rB=45000.

2) In the Angle assignment operation statement the rA can't be omitted, otherwise alarm; it can be written rN=1*sine (rB).

3) In the PROGRAM EDITION working mode, all the special expressions can be get by pressing the hp1 2 key in the plate.

4) Each assignment statement only can has one operator (except the multiply operation first, dividing operation later)

Correct example: r001= r002 + r003 – r004.

Incorrect writing: r001 = r002 + r003

r001 = r001 – r004

【Example】

```
r3=r2*r41          ;  the result of r2 multiplying r41 gives to r3
r2=sqr (100)       ;  the date of 100 draw gives to r2
r1=0               ;  the initialization r1 is Angle 0°
r51=1000*sin(r1)   ;  the sine value of Angle r1 multiplying 1000 gives to r51
r1=r1+1000         ;  the Angle adds 1°
```

7.2.2 Conditional Statement

The "if" conditional statement can be formed a branch structure. It makes judgment according to the given condition, in order that it decides to execute which branch block.

The basic format of if condition is:

If (relationship expression) then statement 1 else statement 2

Its meaning is: if the expression exists, execute the statement 1 behind the then, otherwise execute the statement 2 behind the else.

【Explanation】

1) In the relation expression, the relation operation has **<**, >, =; (less than, more than, equal to); such as : rA< rB, rA> rB, rA= rB;

2) The statement 1 and statement 2 can be transfer statement P, calling statement H, or assignment statement, but only can be one of them;

3) The transfer symbol P and calling symbol H

4) The operator in assignment statement only can be +, - (addition, subtraction); such as rN=rC + rD, rN=rC – rD.

【The basic format and signification of if statement:】

```
if (rA= rB)   then   P1   else   P2                ; if rA=rB, execute P1, otherwise execute P2;
if (rA = rB)   then   P1   else   rN=rC + rD       ; if rA=rB, execute P1, otherwise execute rN=rC+rD
```

if (rA = rB) then rN=rC + rD else P2 ; if rA=rB, execute rN=rC+rD, otherwise execute P2;

Among them:

rN： Variable name;

rA, rB, rC, rD： Variable name, constant too;

P1, P2： Transferred block line number;

H1, H2： The called block line number (in the above format, the place which is transferred by P also can use call).

7.2.3 Statement Program Example

【Program example 1】

In the following program, it mainly demonstrates the method of automatically modifying tool compensation in the program. In the batch machining, suppose each machining 20 piece, the tool T11 will fray one thread in the X direction regularly; when the workpiece recorder is the integer double, the program will make the tool compensation modification automatically.

```
N0010 G00 Z200 X100
……
N1000 r1=r4001 % 20          ; get the remainder of workpiece recorder dividing 20
N1010 if (r1=0) then P1030   ;if the remainder is zero, it will be the integer double of 20
N1020 M20
N1030 r6201=r6201-10         ; No.1 X tool offset reduce one thread
N1040 M20                    ; recycle
```

【Program Example 2】

In the following program, it mainly demonstrates the method of temporary press interference and call in the program. Suppose the pin of UO31 and UO32 control a . In the batch cycle machining, when the handlers want to make the random sample to the workpiece, press G key, the loader collects a sample in automatically.

```
%104
N0010 G00 Z200 X100          ;
N0020 G00 Z180 X80           ;
N0030 G01 Z100 F200          ; cut outer circle
N1000 if (r4009=71) then H2000;If “G” key is pressed in forestall, then call program with H2000
                               as its head block number
N1010 G01 X0 F200            ; cut off
N1020 r2031=1                ; loader returns to original position
N1030 r2032=1                ;
N1040 M20
N2000 r2031=0                ; loader goes forward
N2010 r2032=0                ;
N2010 r4009=0                ; clear “G” state
N2020 M99
```

【Program example 3】

In the following program, it mainly demonstrates the function operation. According to the elliptic parameter equation finger out that the point in ellipse are moving along the ellipse path by short line

approaching method.

The parameter equation of ellipse is X=50*sin(a)，Z=100*cos(a); because the X is diameter program, so X=100*sin(a).

```
%103
N0010 G00 Z0 X0
N0020 G00 Z200 X200
N0022 G00 Z190 X110              ; confirm the start point of G01 path
N0030 r1=0000                    ; the initial value of r1 is 0°
N0040 r12=100000*sin(r1)         ;100000 the angle sine value multiplies 100000
N0050 r13=100000*cos(r1)         ;100000 the angle cosine value multiplies 100000
N0060 r12=r12-110000             ; finger out the relative difference of ellipse start point path
                                   and G01 start point
N0070 r13=r13-190000
N1000 r2=100000*sin(r1)          ;The angel sine value multiply 100000 giving to r2
N1010 r3=100000*cos(r1)          ;The angel cosine value multiply 100000 giving to r3
N1020 r2 = r2 - r12
N1030 r3 = r3 - r13
N1040 G1 Xr2 Zr3 F500            ; Moving along the ellipse path
N1050 if (r1=360000) then P1080  ;If the angel equals to 360°, then exit
N1060 r1=r1+1000                 ; the value of r1 pluses 1000 (angel pluses 1)
N1070 M97 P1000                  ; cycle
N1080 M02
```

7.3 Process Monitoring and Execution

The common part program is executed according to the designed path one by one in advance, it can't make the prompt response for the real time happened matter in the command executive midway.

In the system, there are ten inspectors which are numbered according to 0# to 9# turn, called 0# inspector, 1# inspector… 9# inspector. Their work is called process monitor; in the same time of executing the common part program, they specially track with the matters which are real time happened in the part program execution process. But the user must describe the task of the inspector, and tell them to track what matter, how to deal with it after it happen. The inspector without task is in dormancy state.

Each inspector has a monitor register and a monitor management register separately, the corresponding turn number is as following:

Inspector number: 0#~9# number in turn

Monitor register number: r7000~r7009 corresponding the inspector number in turn.

Monitor management register number: r7010~ r7019 corresponding the inspector number in turn.

In the progress of machining, if there are progress monitors enabled, on the right side of the program status bar is displayed "M-n". Thereinto, M indicates the process monitor, and n indicates the number of enabled process monitors. The operator can also view the states of process monitors being used on the macro variable window.

E.g. M-2 indicates there are two process monitors enabled.

7.3.1 Process Monitor Description (r7000)

Using the process monitor, at first must describe the process monitor according to the rule. The process monitor description includes a assignment statement and a if condition statement, neither of the two can be dispensed; the assignment statement indicates the monitor object, the if condition statement can make the judgment according to the information by monitor, and decide to execute some branch block or not.

【Description method of process monitor】

1) Assignment description:

What is called assignment description, is to designate the monitor object by a assignment statement; all assignment statements which aim at monitor register(r7000~r7009) are called monitor description statement.

For example: r7000= r1001 or r1002; 0# monitor r7000 object are interface input signal UI01, UI02
r7001=r4003(rotate speed); 1# monitor r7001 comes from spindle real time speed

2) Condition description

What is called condition description is to designate how to make judgment according to the monitored information;

The condition description statement and common conditional statement have the unanimous basic form, and the program rule is basic same.

If (relation expression) then statement 1 else statement 2;

Among them: in the condition description statement, the left of if related expression is monitor register name;

In the condition description statement, the call can't be used in then, else, only assignment or transfer.

For example: if (r7000=1) then P1500 ; if the input signal can meet the requirement, then turns p1500

If (r7001>6000) then r3=r3+1 ; meet the requirement, r3 adds 1

【**Explanation**】

1) The system regulation: all assignment statement of aiming at r7000~r7009, are the description to monitor 0#~9#; according to the program execution path, the latest description is effective.

2) The operator of assignment description statement only can be one of"+", "-", "or", "and"; otherwise alarm.

3) the conditional judge statement is allowed to make the assignment description, such as if (r5004=3) then r7000=r1001 + r1002，it is the legal statement.

4) system rule: if the r7000～r7009 appears in the left conditional expression of if conditional statement, it is to say that's the conditional description to 0#～9# monitor. The call can't be used in then, else, only assignment or transfer.

5) The descriptions both assignment and "if" condition should be generated in pair; if there is no

assignment description, or evaluate a non-monitoring variable, then r7000=0; if there is no if conditional description, it's allowed but with little signification and without monitor function.

6) In the if condition description, the system can't execute the description statement immediately, but wait for the corresponding serial number monitor function opened, then starts the monitor.

7) If the assignment meets the requirement, it's not the conditional transfer behind, but the common variable evaluated, such as if (r7000=100) then r3=r3+1，the monitor won't close.

7.3.2 Start and Close of Process Monitor

After making the process monitor rule description, evaluate the monitor management register (r7010~r7019), then can open or close the monitor. There are following working modes for opening the monitor.

Assignment	Process mode of matter
0	Close the opened monitor, let the inspector in the dormancy state; keep its task, can open again and go on the monitor;
1	Open the monitor; in the monitor process, make the judgment according to the matter's relation expression, if the condition comes into existence, the process mode is as following: 1) If the inspector execute the assignment on common variable or output variable, it won't impact the execution of common program command, go on the monitor; 2) If the inspector wants to execute the special assignment, then close the monitor, stop the current motion command immediately, wait for the execution ending of all executing MST command, execute the special assignment; 3) If the inspector want to execute transfer, then close the monitor, stop the current motion command immediately, wait for the execution ending of all executing MST command, execute the transfer;
2	Open the monitor; in the monitor process, make the judgment according to the matter's relation expression, if the condition comes into existence, the process mode is as following: 1) If the inspector execute the assignment on common variable or output variable, it won't impact the execution of common program command, go on the monitor; 2) If the inspector wants to execute the special assignment, then close the monitor, stop the current motion command immediately, wait for the execution ending of all executing MST command, execute the special assignment; 3) If the inspector want to execute transfer, then close the monitor, stop the current motion command immediately, wait for the execution ending of all executing MST command, execute the transfer;
Note: In execution, when meet the M20/M02, Whether closes the monitoring is related with the parameter P419_d5 bit; if it is set to 0, then close the monitoring, clear the task description. Whether closes the monitoring is related with the parameter P419_d4 when resetting. Special assignment means to evaluate the program control register r5008, system special variable set 1 and set 2.	

【Example】

```
r7010= 0      ;   if the system writes zero to r7010, then cancel the r7010 monitor;
r7016= 1      ;   open r7016 monitor; if the system meets the condition, stop the current
                  motion command immediately, execute the assignment or transfer;
r7012= 2      ;   open r7012 monitor; if the system meets the condition, executes the
                  valuation or transfer after executed current motion command;
```

【Explanation】

1) Once opened monitor, in the same time of executing the common part program, in fact there are two description statements are alternated ceaseless, executed circularly in the system

inner; until they are closed.

2) The “if” statement opens or closes the process monitor are allowed; such as if (r1=0) then r7010=1.

3) It’s allowed that one monitor opens another one, but won’t closed itself; such as if (r7001=0) then r7010=1, 1# monitor when it meets the requirement opens 0# monitor, at this time 1# is still open.

4) In making program, before open the monitor, it must make the description to the monitor; if it opens a monitor (without task) which don’t have description, the system will alarm prompting “E400: process monitoring variable undescribed”.

5) If opened some monitor, change its monitor description in the instance of don’t meet requirement and don’t stop monitor, then the new monitor description will change the old description, and the monitor will still on the opening state, needn’t restart. But it’s very dangerous in this state, the monitor must be closed at first, then change its monitor description, then open it again.

6) After meets the monitor requirement, if the executed command in monitor condition is skip command (Pxxxx), close the monitor automatically; when the executed command is assignment or calculate command, don’t close the monitor.

7) Monitoring administration register can be regarded as condition judgment, for example: if (r7010 > 1) then (r002 = 2)

8) If the monitor is opened, under the PAUSE/Block stop /Cycle stop state, the monitor is still effective; its assignment or operation command will be executed after meeting the requirement; the skip command can be executed by pressing CYCLE START key.

9) Once the monitor is opened, in the system inner, because its two description statements are alternated ceaseless, executed circularly, its execution speed is faster than the common program; for the calculation statement which executes assignment after meet the requirement, it may be executed for time after time, this variable result is uncertainly(suggest to don’t use it like this). Pay attention in the common part program, this variable must be used carefully.

For example, if (r7000=100) then r3=r3+1; // if meet the condition, r3 pluses 1, at this time pay much attention, the value of r3 is very uncertainly (the accumulation frequency is very high), if the call or transfer of subprogram in the program uses r3, it will induce the program execution uncertainly, suggest that the uncertainly variable like r3 can’t be used for the real function of all command in this instance.

10) In the common program, read the monitor register, monitor management register, it’s value is zero or uncertain; such as r5= r7000, then the value of r5 is uncertain.

11) If one monitor is open, it can check the signal change above 3ms width; when ten monitor are open, they can check the signal change above 30ms width.

7.3.3 Monitor Program Example

【Program example 1】

The following programs are introduced the explanation of process monitor.

In the batch machining, when the rough bar uneven length, it's hard to confirm the machining start point. Suppose a sensor is installed in the tool, then the system can use the sensor to confirm the machining start point.

In the following program, if UI05 corresponding pin is connected to the system detection signal; when the tool is far away the workpiece UI05="1"，when the tool is near the workpiece UI05="0"，when the tool is moving to the direction of near workpiece from far away, it is stopped at the position UI05="0", and confirm this point to be the machining start point.

```
%105
 N0010 r7000=r1005        ; 0# monitor tests the state of input signal r1005, it forms the
                                monitor description with nether judgment statement
 N0020 if(r7000=0)  then  P1000;If the signal r1005 is monitored to be "Zero", then turn to P1000
                                program running
 N0030 G00 Z300 X200      ; At first orientate to the initial point
 N0040 G00 X180           ;
 N0050 r7010=1            ; Opening 0# monitor by mode 1, when it meets the requirement,
                                stops the motion immediately, and turn P1000
 N0060 G00 W-100  F1000   ;The tool is closing the workpiece from far away
 N0070 r7010=0            ; If the last command is finished, but nothing can be checked,
                                then close the 0# monitor
 N0080 G00 Z300 X200      ; Back to initial point
 N0080 M02
 N1000 G50 Z200 X180      ; Set this point to be the machining start point, set the workpiece
                                coordinate system again
 N1020 G01 Z90  F500      ;Cut outer circle
 N1030 G01 X0             ; Cut off
 N1040 G51                ; Revert the workpiece coordinate system
 N1050 G00 Z300 X200      ; Back to initial point
 N1060 M02                ;
```

【Program Example 2】

The following programs are introduced the explanation of process monitor.

The system has exterior cycle start/pause function, the handlers stand far away the system and can use the exterior press-button to come true the start/pause function. But in the position which is far away the system, it's hard to come true "single block stop". In the following program, it will come true the function through the process monitor

In the program, if UI05 corresponding pin is connected to the system detection signal; when press the switch UI05="0", when release UI05="1". After opened the monitor, this press-button can make the cycle switch between single block and continuum execution working mode.

```
%106
 N0030   r7000 =r1005                ; 0# monitor tests the state of input signal r1005
 N0040   if(r7000 =0) then r5008=2   ; If the signal r1005 is monitored to be "zero", then pause
 N0030   r7001 =r1005                ; 1# monitor tests the state of input signal r1005, r1006
```

```
N0040   if(r7001 =1) then r7010=2 ;If the press-button is released, then open the 0# monitor again
N0070   r7011=2                   ; Open the 1# monitor by mode 2
N0070   r7010=2                   ; Open the 0# monitor by mode 2
N0010   G0 X100 Z200
N0020   G00 X50 Z190
N0050   G1 X50 Z160 F300
N0080   G1 U10 W-20
N0080   G1 W-20
N0080   G1 U10 W-20
N0090   G0 X100 Z200
N0160   M02
```

【Program Example 3】

The following programs are introduced the explanation of process monitor.

In the following program, if the UI05, UI06 pin are connected to detection signal, output signal UO05 pin, in the machining process, when UI05, UI06 are “0”, the system is required to output “0” from UO05 pin, and output “1” in other instance.

```
%107
 N0010 r7000=r1005 or r1006       ; 0# monitor tests the state of input signal r1005, r1006
 N0020 if(r7000=0) then r2005=0   ;If two signals r1005, r1006 are “zero” at the
                                    same time, UO05 outputs “zero”;
 N0030 r7010=1            ; Opening 0# monitor by mode 1, when it meets the requirement,
                            UO05 outputs “zero”;
 N1000 G00 Z300 X200      ; Orientate to the initial point
 N1010 G00 X180           ;
 N1020 G01 Z90   F500     ;Cut outer circle
 N1030 G01 X0             ; Cut off
 N1050 G00 Z300 X200      ; Back to initial point
 N1060 M97 P1000          ;
 N1070 M02                ;
```

7.3.4 Multi-statement Global Monitoring Function

The execution function of multi-statement global monitoring is mainly used for controlling the feeder machine. It can be carried out the feed preparation instead of wasting time while the workpiece is machined, so that the working efficiency is enhanced. The new version is not only performed the more complicated global monitoring program, but also increased several subsequent execution statements with “>”. The following program are described the usage of the multi-statement global monitoring.

It is supposed that the system will output UO05/UO09 signal to control the preparation work before the feeder is completed the feeding when it is detected the UI05 signal.

```
%001
 N0010 r7000 = r1005 ;          Input signal state of r1005 by 0# monitoring inspection
 N0020 if(r7000 = 0)then(r2005 = 0) ；  If monitoring r1005 is “0”, then UO05 outputs “0”
 N0030 > r5008 = 101000      ;Delay 1 second；
 N0040 > r2009 = 0           ; UO09 outputs “0”
 N0050 > r5008 = 100500      ;Delay 500 millisecond；
 N0060 > r2005 = 1           ; UO05 outputs “1”
```

```
N0070 > r5008 = 100300      ;Delay 300 millisecond；
N0080 > r2009 = 1           ;UO09 outputs “1”；
N0090 > r7010 = 0           ; Close 0# monitoring

N0100 r7010 = 1             ; Open 0# monitoring by method 1
N0200 G0 X100 Z100
N0210 G0 X50 Z50            ; Positioning to the initial point
N0220 G1 X48
N0230 G1 Z0 F100
N0240 G1 X2 F100
N0250 G0 X100 Z100          ; Return to the position point
N0260 M30
```

N0010 — N0090 are the monitoring description statement, which is depicts the signal control procedure of the signal control; N0200— N0260 are the main programmings of the machining workpiece; N0100 will automatically carry out the feed preparation as long as the r1005 signal becomes to “0” during performing the main programming after it opens the 0# monitoring.

[Explanation]

1) This function is the extension for the process monitoring, and the process monitoring function is regular and never changed. The difference between the process monitoring and the multi-statement global monitoring is that the former performs only one monitoring and the later performs more.

2) During the monitoring subsequent commands, use >r5008 = 100000~199999 to carry out the delay of the 0~99.999 second; and the front of “1” is delay mark.

3) If the main programming is controlled and then turn to execute the N1000, it will write as “>P1000” in the following statements.

4) If the main programming has been performed (when it meets the M02 M30 M20), it is related to the parameter P419_d5 bit whether closes the monitoring: if it is set to 0, close it, and clear the task description.

5) User should waiting from the completion of the monitoring even the end program is controlled; but the resetting (P419_d4=0) and ESP does not need to wait, till the end monitoring statement is performed.

7.3.5 Pulse Monitoring (r7100)

In this system, there are other four pulse monitor variable with number 0#～3# in turn. They specially engage the pulse count of input signal. Each pulse monitor variable has a monitoring register and a management register separately, the corresponding number in turn are as following:

Pulse monitoring variable number: 0#～3# number in turn

Pulse monitoring register number: r7100～r7103 corresponding the monitor variable number in turn

Pulse monitoring management register number: r7110～r7113 corresponding the monitor variable number in turn

What is called pulse monitor, it belongs to process monitor too, its description mode, open/close mode; all rules are absolutely same to process monitor.

In the machining process, when the pulse monitoring is started, the right of process status bar will display “P-n”. it also can check the state of system current used pulse monitoring. Such as: P-1, expresses one pulse monitoring is started.

◆The explanation of pulse monitoring:

1) At first make the description to pulse monitoring variable; (all assignment statement for r7100 are description)

Such as: r7100 = r1001; r7100 adds 1 for the pulse signal of interface UI01

2) Then make the description to pulse monitoring condition;

Such as: if (r7100>50) then P1500 ; if the r7100 count value is greater than fifty, then turn P1500

3) And then start the monitor through the assignment of pulse monitoring manager

Through the assignment of pulse monitoring management register(r7110～r7113) , can open or close the monitor.

Among them: when the assignment 0, 1, 2 are same to process monitor.

When the assignment is 5, the pulse monitor counter is cleared to 0.

When the assignment is 6 or 7, the operator should set the pulse monitor count mode.

Assignment	Troubleshootings
0	Close the opened monitor, just the same to process monitor
1	Open the monitor; the process method is absolutely same to process monitor
2	Open the monitor; the process method is absolutely same to process monitor
5	When the assignment is 5, the pulse monitor counter is cleared to 0.
6	Set the counter mode: when the calculation result is changed into0 from non-zero, the inner counter will plus 1.
7	Set the counting mode: when the calculation result is changed into non-zero from 0, the inner counter will plus 1.

◆How to count the pulse monitoring ;

1) When the pulse monitoring is opened, it can’t execute the assignment description statement repeatedly, when the last operation result compared with this time, it matches the following skip, and the inner counter will add 1.

If set 6, the last calculation result is non-zero, however this time operation result is 0, the inner counter will plus 1.

If set 7, the last calculation result is 0, however this time operation result is non-zero, the inner counter will plus 1.

2) In the initial that the pulse monitoring is opened, the counter is automatically cleared to zero.

3) When only one pulse monitoring is opened, it can check the above 3ms width signal change; when four pulse monitoring are opened, it can check the above 12ms width signal change.

7.3.6 Pulse Monitoring Program Example

【Program example】

The following programs introduce the use of pulse monitoring.

In the following program, if UI05 corresponding pin is connected to the system detection signal; in the machining, when the times of UI05 changing from “0” to “1” is greater than ten, end the program.

```
%108
 N0010 r7100 = r1005          ; Making the plus 1 count to r1005 pulse signal
 N0020 if(r7100>10)  then  P2000;If the monitor pulse number is greater than 10, then
                                turn to P2000 running
 N0030 r1=0                   ;
 N0040 r7110=7                ; Set the count mode, when it is changed from zero to non-zero,
                                the inner counter will plus 1.
 N0050 r7110=2                ; Choosing the monitor working mode 2 to open
 N1000 G00 Z300 X200          ; Orientating to initial point
 N1010 G00 X180               ;
 N1020 G01 Z90   F500         ; Cut outer circle
 N1030 G01 X0                 ; Cut off
 N1050 G00 Z300 X200          ; Back to the initial point
 N1060 M97 P1000              ;
 N2000 M02                    ;
```

7.3.7 Variable Transfer Register (r7900)

Variable transfer register: r7900; (the system forbid to use this variable in the process monitor description statement)

[Explanation]

1) When r7900 is set to 0 or 999: The overall common variables in the monitoring backup area are copied to the common variable storage area;

2) When r7900 is set to 1~199: The common variable in the monitoring backup area of this current numerical value copies to the common variable (it separately corresponds to the r1~r199);

3) When r7900 is set to -999: The overall common variables in its storage area are copied to the monitoring backup area;

4) When r7900 is set to -1~ -199: the common variables of the current numerical value are copied to the corresponding common one in the monitoring backup area (it separately corresponds to r1~r199).

【Use example】

```
r7900= 0      ;
```

Write any 0 to this register, it expresses all common variable in monitor backup area are copied to common variable storage, the following will explain this register function.

In an ordinary way, in order to avoid the collision, when some common variable is used in the monitor, this common variable must be avoided using in the common block. Because the system has a common variable storage, and a backup storage of monitor variable, the two section variable value

change may not synchronous.

Example as common variable 3, explain the change of r3:

1) When r3 assignment statement is executed, r3 is modified in the common program, then the r3 in monitor will be modified immediately too;

2) When the r3 assignment statement is executed in the monitor, it only has modified the r3 in backup monitor; now the r3 in common program isn't modified.

3) When the variable transfer statement (r7900=3) has been executed. The system will copy the single common variable r3 in monitoring backup area to the common variable storage area; in this case, r3 in the common programming equals to the one in the monitoring.

【Program Example】

In the following program, if UI05 corresponding pin is connected to the system detection signal; in the machining, when the times of UI05 changing from "0" to "1" is greater than ten, after the cycle execution is finished, the program end.

```
%109
N0010 r7100 = r1005      ; Making the plus 1 count to r1005 pulse signal
N0020 if(r7100>10) then   r1=1   ;If the pulse number is greater than ten, r1 equals
                                    to 1, in fact it has modified the r1 in backup monitor
N0030 r1=0                ; Set the initial value
N0040 r7110=7;Set the count mode, change from zero to non-zero, the inner counter will plus one.
N0050 r7110=2             ; Choose the monitor working mode 2 to open
N1000 G00 Z300 X200       ; Oriented the initial point
N1010 G00 X180            ;
N1020 G01 Z90   F500      ; Cut outer circle
N1030 G01 X0              ; Cut off
N1050 G00 Z300 X200       ; Back to the initial point
N0060 r7900=1; Read the common variable value r1in the monitoring backup area return
N0020 if(r1=1) then   P2000;If r1 equals to 1, then turn to P2000

N1060 M97 P1000           ;
N2000 M02                 ;
```

7.4 Attached List

7.4.1 ASCIIList

character	ASCII code	character	ASCII code	character	ASCII code	character	ASCII code	character	ASCII code	character	ASCII code
Space	32	0	48	@	64	P	80	`	96	p	112
!	33	1	49	A	65	Q	81	a	97	q	113
"	34	2	50	B	66	R	82	b	98	r	114
#	35	3	51	C	67	S	83	c	99	s	115
$	36	4	52	D	68	T	84	d	100	t	116
%	37	5	53	E	69	U	85	e	101	u	117
&	38	6	54	F	70	V	86	f	102	v	118
,	39	7	55	G	71	W	87	g	103	w	119
(	40	8	56	H	72	X	88	h	104	x	120
)	41	9	57	I	73	Y	89	i	105	y	121
*	42	:	58	J	74	Z	90	j	106	z	122
+	43	;	59	K	75	[	91	K	107	{	123

,	44	<	60	L	76	\	92	l	108	|	124
-	45	=	61	M	77	]	93	m	109	}	125
.	46	>	62	N	78	^	94	n	110	~	126
/	47	?	63	O	79	-	95	o	111		

7.4.2 Often Used Color and Code Value Corresponding List

name	code	name	code	name	code	name	code	name	code	name	code
black	0	cyan	41	purple	85	red	168	yellow	245	grey	251
blue	6	Light blue	54	light green	105	pink	172	golden	248	hoar	253
green	35	green yellow	80	brown	126	orange	231	silvery white	250	white	255

CHAPTER EIGHT CUSTOMIZATION COMMAND PROGRAM

8.1 Customization Command

They system has a set of customization command except the offered standard M command. The machine manufacturer can set some commands to come true the control of additive equipment; however the operator can operate the additive equipment by input the command expediently.

The system has provided 25 customization commands from M50 to M74, how many is need exactly is edited by the machine manufacturer. The customization command must be edited in %254 program, at first debug %254 program successfully in AUTO working mode, then harden the program in EDIT working mode; after the program is hardened successfully, the hardened customization command can be used in AUTO mode or other program(i.e. %000～%253) , otherwise the system will alarm that there is no customization command.

This chapter will express the written customization command method from example:

【Program example:】

It is supposed that the manufacturer install a loader in the machine. UO31 output pin controls the forward and back of the loader, when it is output “0”, the loader goes ahead, when it is output “1”, the loader will go back; checking the loader forward and back is in the place or not by UI05/UI06 inputting pin, when the system has checked the UI05 is “0” the loader is in the forward place, when the system has checked the UI06 is “0” the loader is in the back place; in the example M51 command is used for controlling the loader forward an check if it is in-position ; M52 command is used for controlling the loader backing, and check if it is in the back place.

```
%254
 N0010 M98 P1000    ; No. %245 program starts execution, call the M51command at first
 N0020 G04 D3                ;
 N0030 M98 P2000    ; Call M62 command
 N0040 G04 D3                ;
 N0050 M02                ;

 N1000 –M51                  ; The start sign of M51 command
 N1010 r2031=0                    ;Loader forward
 N1020 if (r1005=1) then P1000    ;If it detects that UI05 is “1”, then start the cycle check
 N1030 M99                        ; M51 command is end

 N2000 –M52                  ; The start sign of M62 command
 N2010 r2031=1                    ;Loader backing
 N2020 if (r1006=1)  then  P2000 ;If it detects that UI06 is “1”, then start the cycle
                                       checking
 N2030 M99                        ; M52 command is end
```

8.1.1 Customization Command Program Format

In No. %254 program, the system customization command program format is: input a set program which starts from “-M50” to “M99” in the program, this block is formed M50 command. The other block

which start by “-M51 ～ -M74”and end by “M99” is formed M51 ～ M74 command.

【Format】

```
N1000 –M51              ; The start sign of M51 command
N1010 r2031=0                   ;Loader forward
N1020 if (r1005=1)  then  P1000      ;If it detects that UI05 is “1”, then start the cycle
                                      checking
N1030 M99                       ; M51 command is end
```

【Explanation】

The customization command is mainly formed by statement, when G code want to be added in it, the position parameter **P404_d2**=1 must be set, it also is to say M51 command allows the G motion code, otherwise the system alarms.

8.2 Customization Command Store (P254)

The customization command harden is in No. %254 program. So No. %254 program also is called the user-defined command storeroom: P254 mainly includes the M50~M74 command which is formed by statement and block, the system will make the special translate and edit in translating and edition.

8.2.1 Format and Debugging of Customization Command Storeroom

【Format】

P254 program format:

```
N0010   M98 P1000       ;   No.%254 program starts execution
N0020   M98 P2000       ;   Call M51 command
N0030   M98 P3000       ;   Call M52 command
N0040   M02
N1000   -M50            ;   the start of M60
…
N1100   M99             ;   the end of M50
N2000   -M51        ;   the start of M51
…                       ;   the user customized M51 content
N2100   M99             ;   the end of M51
N3000   -M52        ;   the start of M52
…                       ;   the user customized M52 content
N3100   M99             ;   the end of M52
```

The debugging steps of customization command storeroom is as following:

1) Edit the user need customization command according to the above customization command storeroom program format

2) Debug the program by single block mode in AUTO working mode

3) After debugged the program, harden the program in EDIT mode, press the key: hp1 5 ; the machine manufacturer confirm if it’s allowed to modify the hardened program and harden again, it also is to set the position parameter **P404_d0**=1.

4) The system prompts that after the P254 translating and harden succeed, press ESC key.

【Explanation】

After the No.%254 translating and harden succeed, it also can input the customization command in P254 in JOG working mode and run, in order to check the correctness of P254.

8.2.2 Explanation of Customized Command Storage

In the JOG working mode, input the customization command directly when it needs to execute the hardened customization command in P254. In editing program the hardened customization command in system P254 can be used, the operation is simple and convenient.

【Using customization command format】

```
%001              ; execute the customization command beside the No.%254 program
N0010 M51         ; execute M51 command
N0020 G04 D3      ;
N0030 M52   ; execute M52 command
N0040 G04 D3      ;
…
…
N0050 M02         ;
```

【Explanation】

1) The M50~M74 can be used in JOG working mode or other program directly only the No.% 254 program has hardened to FLASH, otherwise the system alarm; when the corresponding customization command isn't written in No. % 254 program, but it is used in JOG working mode or other program, the system alarm too.

2) The operation of how to harden the No.%254 program to FLASH, please refer the No.%254 program explanation, with the detail in OPERATION, 4.3 EDIT working modein this explanation notebook.

3) After the No. %254 program is modified, it must be hardened to FLASH again, otherwise the called M50~M74 command in JOG working mode or other program aren't the latest customization command after modified.

4) In AUTO working mode, when the M50~M74 used variable are used in main machining process, must pay attention to the impaction of variable cross-change to the program

5) M50～M74 command must be used separately , such as if (r1>100) then M51 expression will alarm.

【Special attention】

1) When user-defined command is executed in AUTO mode, press CYCLE DWELL, the system will not stop the blocks under user-defined command, rather it will continue the execution after a short pause.

2) When user-defined command is executed in AUTO mode, if SINGLE is valid, SINGLE function is valid for the user-defined command not to the blocks under the command.

3) When user-defined command is executed in MANUAL mode, press CYCLE DWELL, the

user-defined command execution will be paused.

8.2.3 Customized Command Machining Example

【Program example】

The following is the program of realizing the change tool operation by written customization command. M61 changes the first too; M62 changes the second tool; M63 changes the third tool; M64 changes the fourth tool. Note: if tool changing is executed under following programs, M60 user-defined command should be set at first; see Section 3.4.3.2 in CONNECTION for details.

```
%254
 N0000   M98 P50                          ; Call M61, change the T11 tool number
 N0010   M98 P200                         ; Call M62, change the T22 tool number
 N0020   M98 P350                         ; Call M63, change the T33 tool number
 N0030   M98 P500                         ; Call M64, change the T44 tool number
 N0040   M30                              ; Program end
 N0050   -M61                             ; Start to call M61, start to change tool
 N0060   T11                              ; Target tool number
 N0070   if(r1001 = 0) then(P190) else(P80) ; Judge the current tool is consistent to target tool
                                             or not
 N0080   r2011 = 1                        ; Close the tool post CCW rotation
 N0090   r2012 = 0                        ; Tool post CW rotation
 N0100   r4010 = r39                      ; Changing tool time is set to be 30 seconds
 N0110   if(r4010 < 5) then(P650) else(P120); Judge the tool changing time is too long or not
 N0120   if(r1001 = 0) then(P130) else(P110); Judge the tool position signal
 N0130   r2012 = 1                        ; Close the tool post CW rotation
 N0132   r4011 =50                        ; Set to delay 50ms
 N0134   if(r4011<2) then (P140) else(P134); Delay 50ms
 N0140   r2011 = 0                        ; The tool post CCW rotation
 N0150   r4011 = r40                      ; Set the tool post CCW rotation time
 N0160   if(r4011 < 5) then(P170) else(P160)    ; Check the tool post CCW rotation time
 N0170   r2011 = 1                        ; Close the tool post CCW rotation
 N0172   r4010 = 100                      ; Tool post CCW rotation in-position delayed time
 N0174   if(r4010 < 5) then(P180) else(P174)    ;Tool post CCW rotation in-position delayed
 N0180   if(r1001 = 0) then(P190) else(P700)    ; Judge the in-position signal
 N0190   M99                              ; Tool change end
 N0200   -M62                             ; Start to call M62, start the tool change
 N0210   T22                              ; Target tool number
```

```
N0220  if(r1002 = 0) then(P340) else(P230); The current tool is consistent to target tool or not
N0230  r2011 = 1                          ; Close tool post CCW rotation
N0240  r2012 = 0                          ; Tool post CW rotation
N0250  r4010 = r39                        ; Set the tool change time 30 seconds
N0260  if(r4010 < 5) then(P650) else(P270); Judge the tool change time is too long or not
N0270  if(r1002 = 0) then(P280) else(P260); Judge the tool position signal
N0280  r2012 = 1                          ; Close the tool post CW rotation
N0282  r4011 =50                          ; Set to delay 50ms time
N0284  if(r4011<2) then (P290) else(P284) ; Delay 50ms
N0290  r2011 = 0                          ; Tool post CCW rotation
N0300  r4011 = r40                        ; Set the tool post CCW rotation time
N0310  if(r4011 < 5) then(P320) else(P310); Check the tool post CCW rotation time
N0320  r2011 = 0                          ; Close the tool post CCW rotation
N0322  r4010 = 100                        ; Tool post CCW rotation in-position delayed time
N0324  if(r4010 < 5) then(P330) else(P324); Tool post CCW in-position delayed time
N0330  if(r1002 = 0) then(P340) else(P700); Judge the in-position signal
N0340  M99                                ; Change tool end
N0350  -M63                               ; Start to call M63, start the tool changing
N0360  T33                                ; Target tool number
N0370  if(r1003 = 0) then(P490) else(P380); The current tool is consistent to target tool or not
N0380  r2011 = 1                          ; Close the tool post CCW rotation
N0390  r2012 = 0                          ; Tool post CW rotation
N0400  r4010 = r39                        ; Set the tool changing time to 30s
N0410  if(r4010 < 5) then(P650) else(P420) ; Judge the tool change time is too long or not
N0420  if(r1003 = 0) then(P430) else(P410) ;Judge the tool position signal
N0430  r2012 = 1                          ; Close the CW rotation
N0432  r4011 =50                          ; Set to delay 50ms time
N0434  if(r4011<2) then (P440) else(P434) ; Delay 50ms time
N0440  r2011 = 0                          ; Tool post CCW rotation
N0450  r4011 = r40                        ; Set the tool post CCW rotation time
N0460  if(r4011 < 5) then(P470) else(P460); Check the tool post CCW rotation time
N0470  r2011 = 1                          ; Close the tool post CCW rotation
N0472  r4010 = 100                        ;Tool post CCW in-position delayed time
N0474  if(r4010 < 5) then(P480) else(P474);Tool post CCW in-position delayed time
N0480  if(r1003 = 0) then(P490) else(P700); Judge the in-position signal
N0490  M99                                ; Tool change end
```

```
N0500  -M64                                  ; Start to call M64 and execute the tool change
N0510  T44                                   ; Target tool number
N0520  if(r1004 = 0) then(P650) else(P530)   ; The current tool is consistent with target tool
                                               or not
N0530  r2011 = 1                             ; Close the tool post CCW rotation
N0540  r2012 = 0                             ; Tool post clockwise rotates
N0550  r4010 = r39                           ; Set the change tool time to 30s
N0560  if(r4010 < 5) then(P650) else(P570); Judge the change tool time is too long or not
N0570  if(r1004 = 0) then(P580) else(P560); Judge the tool position signal
N0580  r2012 = 1                             ; Close the tool post CW rotation
N0582  r4011 =50                             ; Set to delay 50ms
N0584  if(r4011<2) then (P590) else(P584) ; Delay 50ms
N0590  r2011 = 0                             ; Tool post CCW rotation
N0600  r4011 = r40                           ; Set the tool post CCW rotation time
N0610  if(r4011 < 5) then(P620) else(P610); Check the tool post CCW rotation time
N0620  r2011 = 1                             ; Close tool post CCW rotation time
N0622  r4010 = 100                           ; Tool post CCW rotation in-position delayed time
N0624  if(r4010 < 5) then(P630) else(P624); Tool post(CCW) in-position delayed
N0630  if(r1004 = 0) then(P640) else(P700); Judge the in position signal
N0640  M99                                   ; Tool change end
N0650  r2012 = 1                             ; Close the tool post CW rotation
N0652  M05                                   ; Close spindle
N0660  00 SE335                              ; Check the tool position signal overtime
N0670  r5002 = 1                             ; Open display
N0680  r5002 = 1000                          ; Display alarm
N0690  r5008 = 1                             ; Motion pause
N0692  M00                                   ; Program pause
N0700  01 SE333                              ; Tool selection number alarm
N0710  r5002 = 1                             ; Open display
N0720  r5002 = 1001                          ; Display alarm
N0730  r5008 = 1                             ; Motion pause
N0740  M00                                   ; Program pause
```

8.3. Foot switch in M61 command

◆Foot switch in M61/ M62, M63/ M64, M65/ M66 and M67/M68 commands:

When you need foot switch to control some reverse motion commands such as M10/M11, M79/M78, and M61/M62 can realize it. For example, M61 is programmed as feeding-in, M62 as feeding-back, set the corresponding I/O of foot switch input pin in parameter P533, and connect them correctly. Then, M61/M62 can be executed alternately.

M61/62, M63/64, M65/66 and M67/M6 can be carried out the foot control function, which are separately set the corresponding I/O name of the pin input by the foot switch, and the setting value is the series number of the common signal name. Note that, the corresponding M61 ~ M68 commands should be solidified.

- **Key control M61~M68:**

When the system input interface is not enough, set parameters P533~P536 to 99; in AUTO or MANUAL mode, press numerical keys can control commands M61~M68:

When P533 is 99, key 1 corresponds to M61, key 2 to M62; when P534 is 99, key 3 to M63, key 4 to M64; when P535 is 99, key 5 to M65, key 6 to M66. When P536=99, press key 7, perform M67, then press 8, perform the M68

【Explanations】

1) In MANUAL/AUTO mode, key control function M61 ~ M68can only be used when the system is free or no program is executed. It cannot be used even in SINGLE stop mode and when window popped up.

2) When the system is power-on, M61/M63/M65/M67 is always executed when foot switch is used for the first time.

3) In the process of M61 (M63, M65 or M67) execution, is RESET key is pressed, the system will go to M62 (M64、M66 or M68) when foot switch is press next time.

8.4 The Foot Switch of M51~M58 Commands

Parameter	Signal name	Function explanation	Range
P557	M51I	M51/M52 execution switch	0---32
P558	M53I	M53/M54 execution switch	0---32
P559	M55I	M55/M56 execution switch	0---32
P560	M57I	M57/M58 execution switch	0---32

[Explanation]

Refer to the Foot Switch of M61 Command in Section 10.3 of **PROGRAMMING** for the related execution function of the command. The foot switch of M51~M58 commands is the different from the one

of the M61 command: fail to perform the “button control M51~M58 command”.

8.5 Call M50-M72 Function Before Machining

This system can be output the signal and supplied by external monitoring based upon the 3 essential points “shift the Auto interface from others; shift to other interface from the Auto one, start and end machining”.

The achieved method: firstly, the corresponding M command can be solidified in the No.254 programming, and then modify the parameters P344~P347, the corresponding numbers are consistent with its corresponding M commands. For example: modify P344 to 51, and then call the M51.

[For example]

If P344 (Call before machining) is set to 50, then perform M50 before executing the 1st machining programming each time.

If 345 (Call after machining) is set to 51, then perform M51 before the end command M30/M02/M20 are executed. If the program is retracted in the halfway, it will not be performed.

If P346 (Enter the Auto Call) is set to 52, then shift to the Auto interface each time from others. As long as no alarm occurs, call and execute the M52.

If P347 (Retreat from the Auto Call) is set to 53, then shift to the other interface from the Auto one. As long as no alarm occurs, call and execute the M53.

The range of parameter P344~P347 is 0~255, the system initialization is 0.

8.6 Call M50-M70 Function by Manual 0 Key

The achieved method: firstly, the corresponding M command can be solidified in the No.254 programming, and then modify the parameters P348, the corresponding numbers are consistent with its corresponding M commands. For example: modify P348 to 51, then the M51 can be called by the 0 key in the manual interface.

[Notice]

1) If only the parameter is set instead of solidifying the corresponding self-defined command in the No.254 programming, then skip it.

2) Due to the additional Call is added, and the machining efficiency may be affected after this function is used.

CONNECTION

CHAPTER ONE INTERFACE

1.1 Rear Cover Interface Position Layout

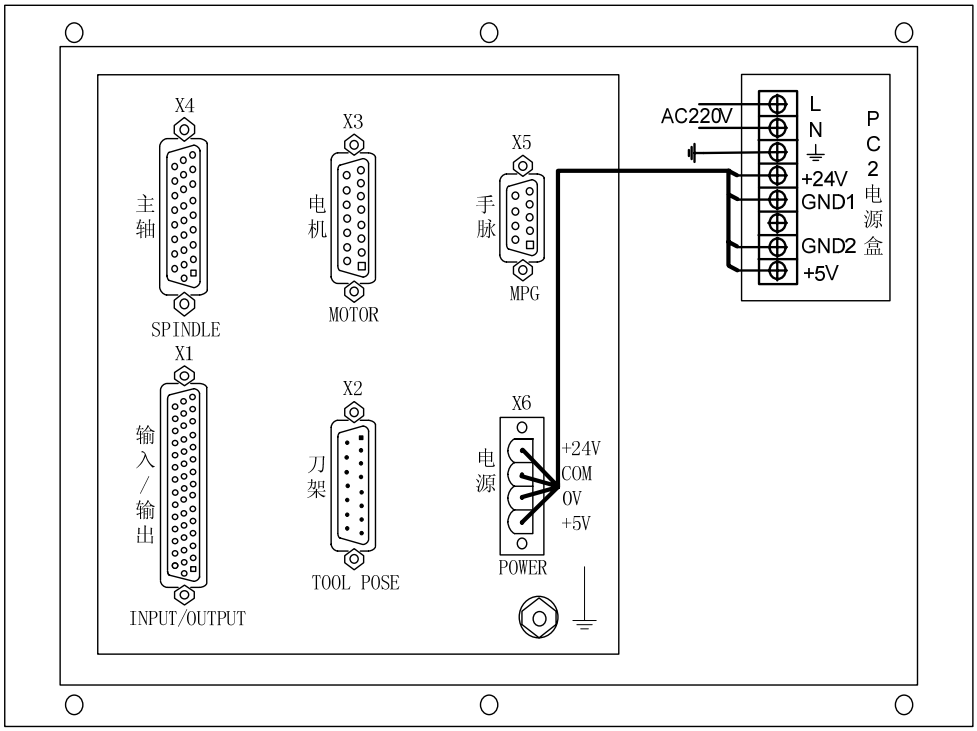


Fig. 1-1 rear cover interface layout

【Explanation】

- X1：Input/output, 44-pin D-type female socket, CNC receives machine signal/CNC signal outputs to machine interface
- X2：Tool pose, 15-pin D-type male socket, input/output interface
- X3：Motor, 5-pin D-type female socket, interface of X/Z drive unit
- X4：Spindle, 26-pin D-type female socket, spindle coder, spindle inverter and Y drive unit interface
- X5：MPG, 9-pin D-type female socket, connect with MPG
- X6：Input power supply, green 4-pin female socket, the power supply box has been connected to the system X6 interface, the user only connects with 220V power supply
- Power supply box: Adopt GSK-PB or GSK-PC2 (their connecting terminals are different); GSK-PB is adopted in Fig. 1-1. It supplies GND, +5V, +24V power.

1.2 Total Frame

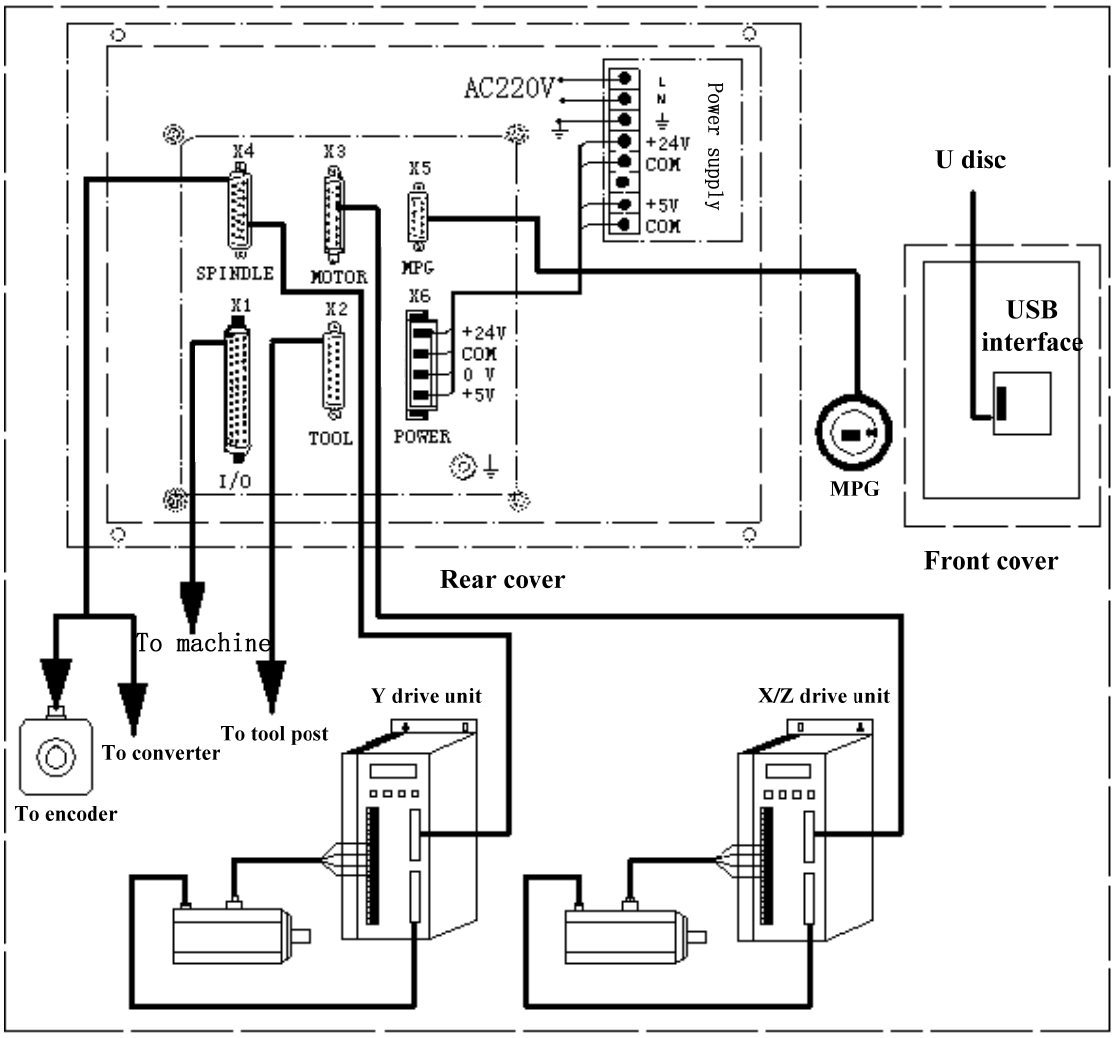
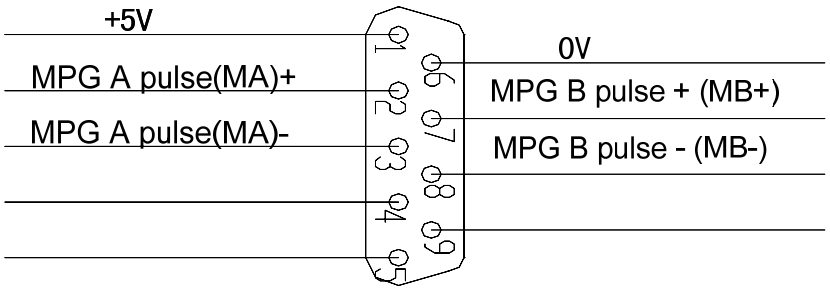


Fig. 1-2 total connection drawing

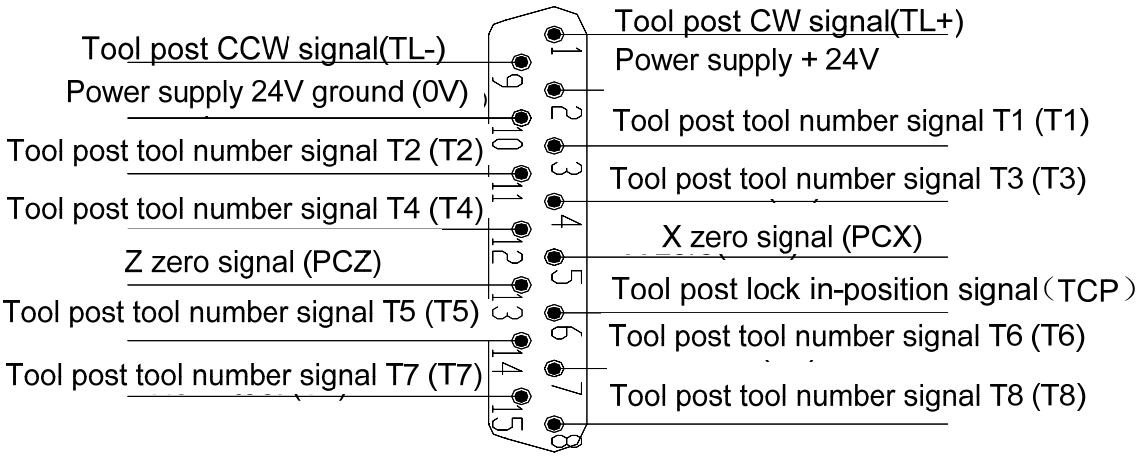
CHAPTER TWO INTERFACE GRAPH

2.1 Interface Table Ⅰ

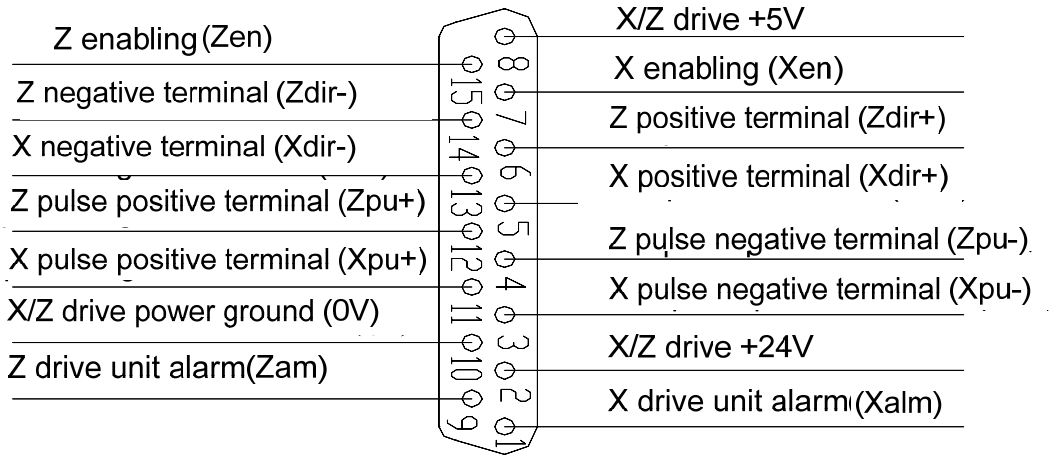
X5 MPG(female)



X2 tool post (male)

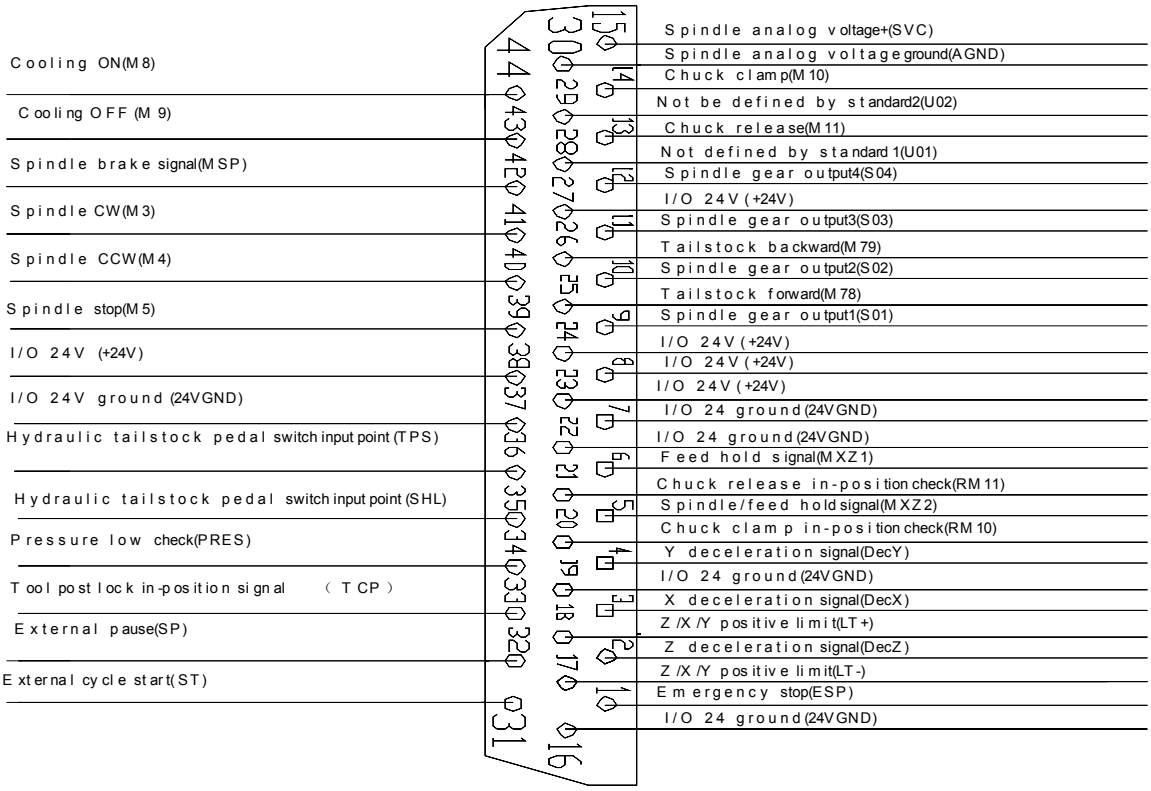


X 3 motor (female)

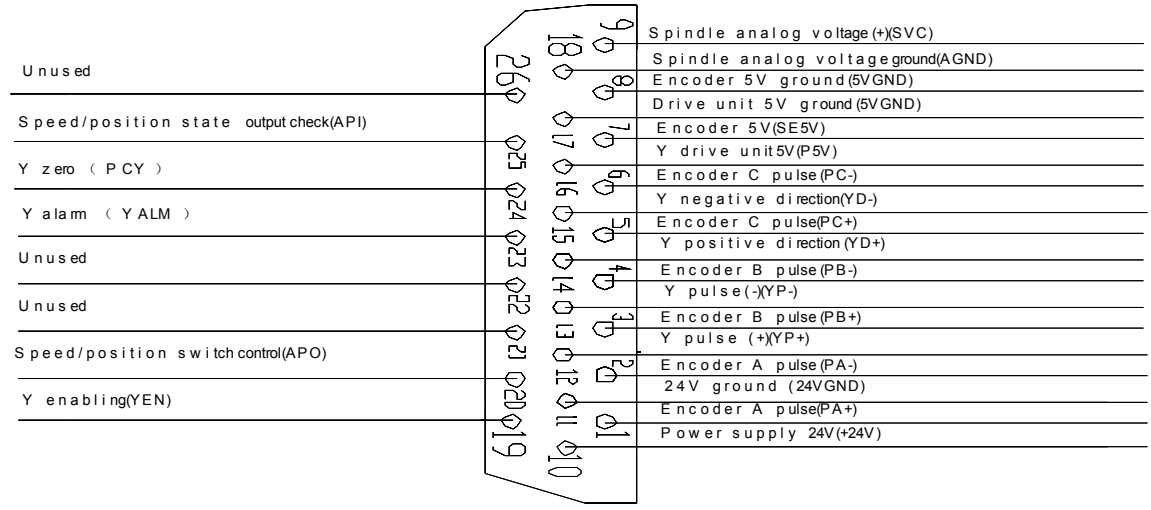


2.2 Interface Table Ⅱ

X1 input/output （female）



X4 spindle (female)



CHAPTER THREE CNC DEVICE CONNECTION

3.1 Front Cover Communication Interface

The CNC system can exchange or transmit data with a U disc through USB interface.

Front USB interface: to connect with U disk.

3.1.1 USB Interface

USB interface is used to connect the CNC system and U disk. The system not only sends the data to the U disk, and also receives the data in the U disk by the USB interface. The U disk is directly inserted into the USB interface on the system panel, and the system automatically identifies and opens the content of the U disk when the U disk creates the file and file name in the root catalog according to the file catalog requirements of the system.

【Note】

1) The system USB interface must be covered without being used and without being exposed for long time, otherwise, which causes the surface metal being oxidated and reduces the interface sensitivity.

2) Do not keeping the U disk in the system USB interface for long time, otherwise, which causes the system interface to be aged, and damages the U disk.

3) After the U disk is used, the user must press ESC to close the U disk before pulling out it, at the moment, the USB icon displayed on the system window disappears and it can be pulled out from the system USB interface, otherwise, which damages the system hardware and the U disk.

3.2 X1, X2 Interface

The CNC system X1, X2 interface have 23-channel digital input, 18-channel digital output, which use the photoelectric isolation.

3.2.1 X1, X2 Interface Signal Definition

Special attentions：

1) The 23-channel digital input signal and 18-channel digital output signal correspond to 23 input pins and 18 output pins. Each pin has their “pin number” and two “signal names”(exclusive signal name and common signal name”. Input general signal names are numbered UI01~UI32, output common signal names are numbered UO01~UO32; among them, UI17~UI25, UO19~UO32 are not led out (unused; displayed in grey in DIAGNOSIS mode). If a signal is not specified, it is displayed in white UI** or UO**, representing it can be used otherwise.

2) This system has already set control and detection pin for most of the devices. These pins are named “standard signal name”, for example, spindle, chuck, tailstock, tool post etc. Set

parameters correctly before used these functions. In DIAGNOSIS mode, the standard signal names are displayed in yellow, representing it cannot be used otherwise.

3) The system has been arranged fixed pins for some commands (such as M91, M21, G31) when it is in the initialization state, that is, the defined interface parameter P500(M21O), P501(M23O), P517(M91I), P518(M93I), P532(G31I). In the diagnosis method, the signal displays with green. The machine factories can connect the circuit based upon the definition of the interface parameter, and the system will detect the corresponding signal according to its pin.

4) For other equipments (Refer to the *Other Option Function* in Section 4.4.9 of **OPERATION**, for example, safety door detection, tri-color indicator control), and the system does not arrange the fixed pin for these functions. The factory can arrange it on any unused pin. Simply speaking, the **serial number** of "**Common Signal Name**" of the unused pin is enabled after registering at the "Interface Parameter" of the parameter working mode. The signal added in the "Interface parameter" column is displayed signal name with green in the diagnosis working mode. For example: if the cutter is the 4-station dynamic tool-post, and then the signal UI05 input is disabled. Set the parameter P511 to 5, which becomes the detection signal (SAGT) of the safety door; when the safety door detection is performed, the system is then inspected the UI05 signal (That is, X2 connects the 14^{th} pin).

5) The value of the input interface signal r1001～r1032 can be read by statement programming; however, not all output signal pins can be read, only the pin "UO" displayed on the diagnosis interface can be performed the statement programming when bit parameter P412_d0=0. If the defined output signal pin is to be programmed which the alarm may occur. For example: variable r1005 is read UI05, that is, the detection of the X2 interface pin number is 14 LEVEL.

6) Both "the No.15 pin of the X1 interface" and "the No.9 pin of the X4"are regarded as signal "spindle analogue voltage positive (SVC)", which are shared with the identical function.

7) Both "the No.30 pin of the X1 interface" and "the No.18 pin of the X4"are regarded as signal "spindle analogue voltage grounding (AGND)", which are shared with the identical function.

8) Both "the No.33 pin of the X1 interface" and "the No.6 pin of the X2"are regarded as signal "tool-post clamping in-position signal (TCP)", which are shared with the identical function.

The definitions of 23-channel digital input signal and 18-channel digital output signal are as follows:

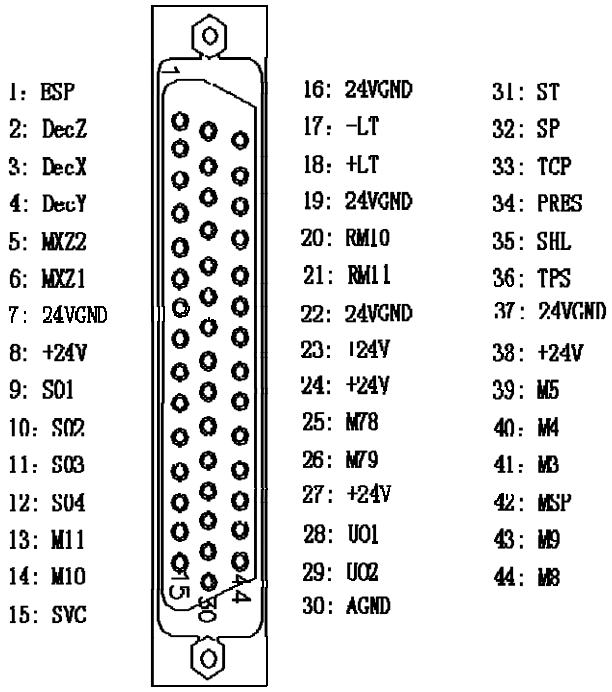
Table 3-1

Interface	Pin	Standard Signal name	General signal name	Variable	Signal function
X1 interface	1	ESP			Emergency stop (emergency stop signal)
	2	DecZ	UI28	r1028	Z axis deceleration signal
	3	DecX	UI29	r1029	X axis deceleration signal
	4	DecY	UI30	r1030	Y axis deceleration signal
	5	MXZ2	UI15	r1015	Spindle//feed hold signal
	6	MXZ1	UI16	r1016	Feed hold signal
	7	0V (24VGND)			IO24V earthing
	8	+24V			24V
	9	S01	UO01	r2001	Spindle gear output 1

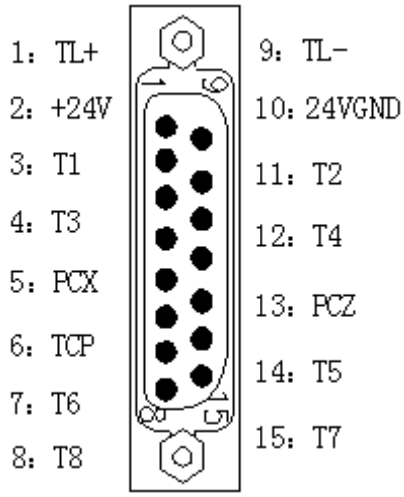
	10	S02	UO02	r2002	Spindle gear output 2
	11	S03	UO03	r2003	Spindle gear output 3
	12	S04	UO04	r2004	Spindle gear output 4
	13	M11	UO13	r2013	Chuck releasing
	14	M10	UO14	r2014	Chuck clamping
	15	SVC			Spindle analog voltage +
	16	0V (24VGND)			IO24V earthing
	17	LT-	UI31	r1031	ZXY negative limit
	18	LT+	UI32	r1032	ZXY positive limit
	19	0V (24VGND)			IO24V earthing
	20	RM10	UI14	r1014	Chuck clamping in-position check
	21	RM11	UI13	r1013	Chuck releasing in-position check
	22	0V (24VGND)			IO24V earthing
	23	+24V			24V
	24	+24V			24V
	25	M78	UO15	r2015	Tailstock going forward
	26	M79	UO16	r2016	Tailstock going backward
	27	+24V			24V
	28	M21O	UO17	r2017	P500 set as user-defined command output
	29	M23O	UO18	r2018	P501 set as user-defined command output
	30	AGND			Spindle analog voltage ground
	31	ST	UI26	r1026	External cycle start
	32	SP	UI27	r1027	External pause
	33	TCP	UI09	r1009	Tool post locked in-position signal
	34	PRES	UI10	r1010	Pressure LOW check
	35	SHL	UI11	r1011	Hydraulic chuck foot switch input
	36	TPS	UI12	r1012	Hydraulic tailstock foot switch input
	37	0V (24VGND)			IO24V earthing
	38	+24V			24V
	39	M5	UO05	r2005	Spindle stop
	40	M4	UO06	r2006	Spindle CCW
	41	M3	UO07	r2007	Spindle CW
	42	MSP	UO08	r2008	Spindle brake signal
	43	M9	UO09	r2009	Cooling OFF
	44	M8	UO10	r2010	Cooling ON

Table 3-2

Interface	Pin name	Standard signal name	General signal name	Variable name	Standard signal name function explanation
X2 interface	1	TL+	UO12	r2012	Tool post(CW) output signal
	2	+24V			Power supply 24V
	3	T1	UI01	r1001	Tool post position signal T1
	4	T3	UI03	r1003	Tool post position signal T3
	5	PCX			X axis zero point signal
	6	TCP	UI09	r1009	Tool post lock in-position signal
	7	T6	UI06	r1006	Tool post position signal T6
	8	T8	UI08	r1008	Tool post position signal T8
	9	TL-	UO11	r2011	Tool post CCW output signal
	10	0V			Power supply 24V ground
	11	T2	UI02	r1002	Tool post position signal T2
	12	T4	UI04	r1004	Tool post position signal T4
	13	PCZ			Z axis zero point signal
	14	T5	UI05	r1005	Tool post position signal T5
	15	T7	UI07	r1007	Tool post position signal T7



X1：input/output, 44-pin D-type female



X2: input/output, 15-pin D-type male

3.2.2 Connection Method of Input Signal

The input signal includes UI01～UI16 ,UI26～UI32 and zero signal (PCX, PCZ, PCY). CNC checks the machine states by the input signal; the signal direction: machine to CNC. The input signal is valid in LOW.

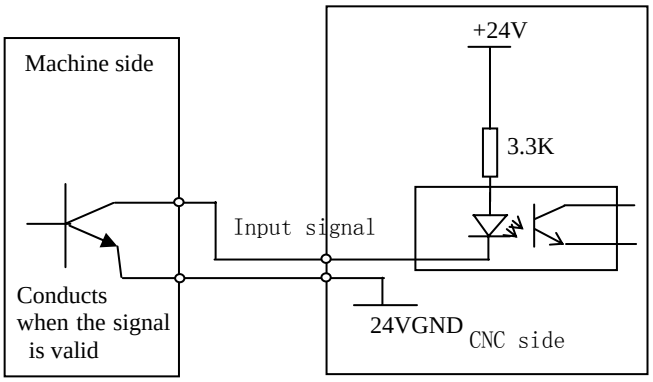
The external input of the input signal: one uses the machine contact switch input, by which the signal is from the machine side press key, the limit switch and relay contact; another uses the electronic proximity switch (transistor) input without the contact.

Note: zero signal (PCX, PCZ, PCY) can connect with one-turn signal of the servo motor (connecting with the servo motor).

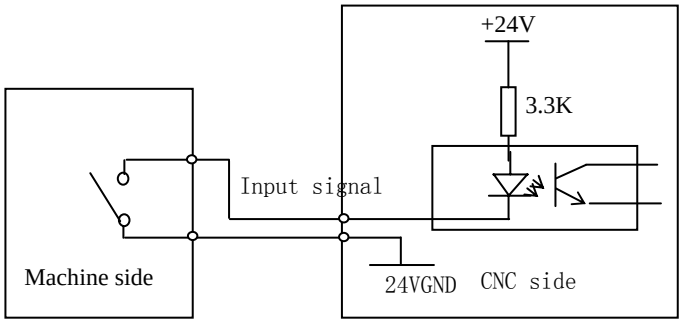
1) The input signal can use the normally-open contact input of the machine contact, and also use

the electronic proximity switch input without the contact (proximity in the output LOW). The interface connection method is as follows:

NPN connection method without contact:



Machine contact connection method:



The contact of the input signal at the machine side should meet the conditions:

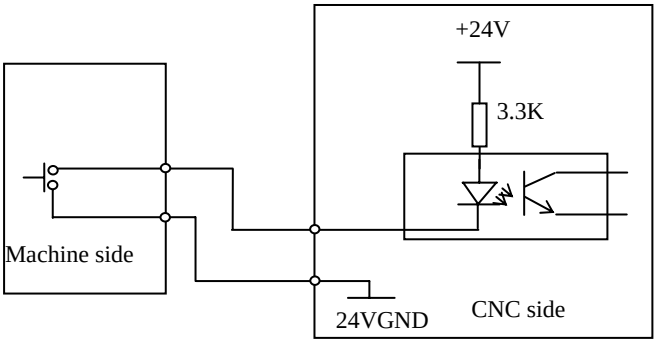
Contact capacity: more than DC28V, 16mA

Leakage current among the contacts in open circuit: below 1mA.

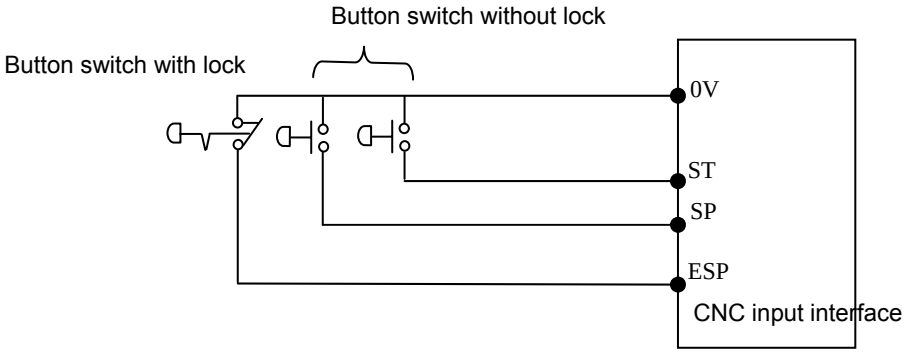
Voltage drop among contact in close-circuit: below 1V (current 8.5mA including the voltage drop).

2) The input signal UI26, UI27 (SP, ST are standard signals) use the normally-open contact of the machine contact switch; ESP uses the normally-closed contact of the machine contact switch with self-lock.

Circuit diagram：



Connection example：



3) Standard signal name definition of input interface

SP：External pause operation key signal.

ST：External cycle start key signal.

ESP：ESP key signal (Note: This signal must be connected) .

SHL(chuck foot switch) ：Foot switch input signal of hydraulic chuck.

TPS(tailstock foot switch) ：Foot switch input signal of hydraulic tailstock.

PRES：Pressure low check, the hydraulic system pressure low check input signal

DecX , DecZ , DecY：Deceleration signal of X/Z/Y axis machine reference point return.

PCX, PCZ, PCY：Zero signal of machine reference point return for of X, Z, Y axis

+LT, -LT：X, Z, Y positive/negative limit switch signals. Connect X, Z positive/negative limit signal to the signal.

Note: +LT, - LT connections are referred to OPERATION, 4.1 Safety Guard.

MXZ1：Feed hold signal. When it is valid (i.e. switch contact ON), the feed is locked (i.e. X, Z stop).

MXZ2：Spindle/feed hold signal. When it is valid (i.e. switch contact ON), the feed is locked (namely, X, Z axes stop). Note: MXZ1, MXZ2 control is referred to **OPERATION, Auto Working Mode**.

RM10：Chuck clamping (outer chuck)/releasing (inner chuck) in-position detection.

RM11：Chuck releasing (outer chuck)/clamping (inner chuck) in-position detection.

Note: the hydraulic chuck and tailstock controls are referred to OPERATION, JOG Working Mode.

T1～T8：Tool post position signal, connecting with the tool post, selecting 4～16 tool selection tool post when the tool selection is more than 8, necessarily, input data in code mode into the CNC system.

TCP：Tool post lock in-position signal connects with tool post.

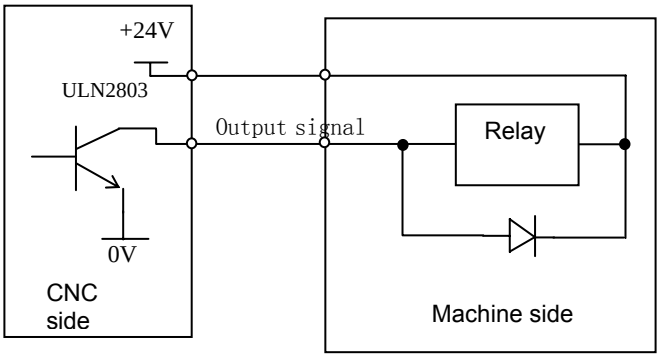
3.2.3 Connection Method of Output Signal

The output signal (UO01～UO18) is used to control the relative operations of the machine, the signal direction: CNC to machine. The signal can drive the relay and the indicator at the machine side. When the output signal is connected with 0V, the output function is valid; when it is not connected with 0V, the output function is invalid. X1, X2 interfaces have 18-channel output interface, and the output signal connection method is as the following figure.

1) output signal connection

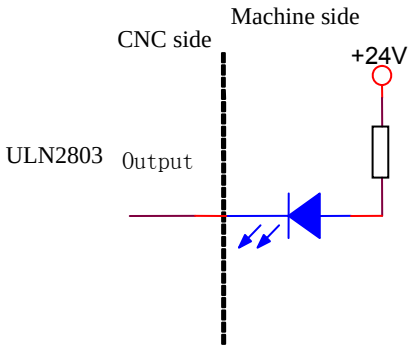
● Drive sensitive load：

Use ULN 2803 output to drive the sensitive load, at the moment, connect with fly-wheel diode near to the coil to protect the output circuit and reduce the interference.



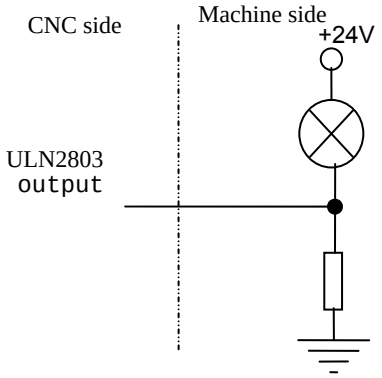
● Drive LED:

Use LUN2803 output to drive LED with a serial resistor to limit the current (about 10Ma) through LED.



● Drive filament indicator

Use ULN2803 output to drive the filament indicator, connect externally one preheat resistor to reduce the current impact, and the preheat resistor value is referred to the condition that the indicator does not light.



2) OUTPUT STANDARD SIGNAL

● The output standard signal are driven by ULN2803 transistor, max. load instant current 200mA.

When the signal is enabled, the transistor is connected and the public terminal is +24V.

- The machine side is connected with the relay and other inductive load, the spark suppressor which is near to 20cm must be used. The serial current-limiting resistance must be used when the machine side is connected with the capacitance load.
- The output standard signal M8, M9, M3, M4, M5, M10, M11, M78, M79 can be set to the level control mode or pulse control mode by the bit parameter, M4, MSP standard signal is set by the parameter to execute the output; the time sequence of the signal output is referred to **OPERATION, JOG Working Mode.**
- The output standard signal S01, S02, S03, S04, M41, M42, M43, M44 gear shifting execution process and time sequence of signal output are referred to **OPERATION, JOG Working Mode.**
- Output standard signal TL+, TL- output to control the tool post CW/CCW.

3.2.4 Input/Output Signal Technical Specification

- Voltage +24V
- Max. load instant current of output signal is 200mA

【Notice】

1) The output voltage should be within 1V when the transistor of electric switch is conducted; the output voltage should be more than 23V when the transistor is turn off.

2) When the input function is valid, the input signal is connected with 24VGND; when it is invalid, the signal is not connect with 24VGND.

3) When the output function is valid, the signal is connected with 24VGND; when it is invalid, the signal is in high-impedance state.

It is suggested that the input/output cable should use the shield cable, and the shield layer is connected with the plug metal cover and the machine to improve the anti-interference ability.

3.3 Machine Zero Return Function and Connection

Relative interface signal of machine zero return is as the following table X1, X2, X4 interfaces:

1) X1 interface：

Pin 2	DECZ	Z deceleration signal
Pin 3	DECX	X deceleration signal
Pin 4	DECY	Y deceleration signal

2) X2 interface：

Pin 5	PCX	X zero signal
Pin 13	PCZ	Z zero signal

3) X4 interface：

Pin 24	PCY	Y zero signal

Bit parameter **P406_d7,P406_d6,P406_d5,P406_d4,P406_d3,P406_d2** set the machine zero return mode.

Bit parameter setting 0 means there is no deceleration signal or zero signal; bit parameter setting 1 means there is the deceleration signal or zero signal.

There are four zero return methods, and their detailed parameter setting methods are referred to *Operation Parameter Working Mode*; the detailed zero return execution process is referred to *Operation JOG Working Mode*.

Connection example：

1) The followings are the two kinds of connection examples of deceleration signal: travel switch, NPN proximity switch; it is suggested that the user should use the travel switch; the concrete connection example is as follows:

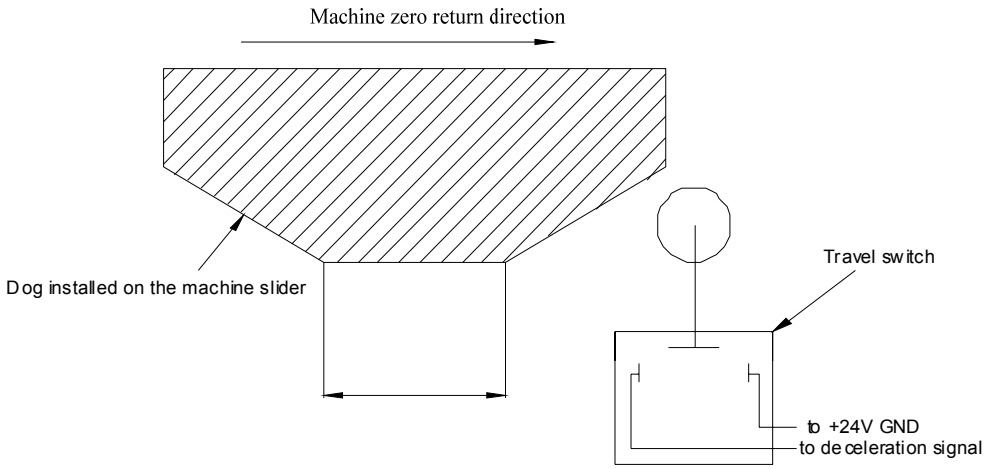


Fig. 3-1 Deceleration signal to the travel switch

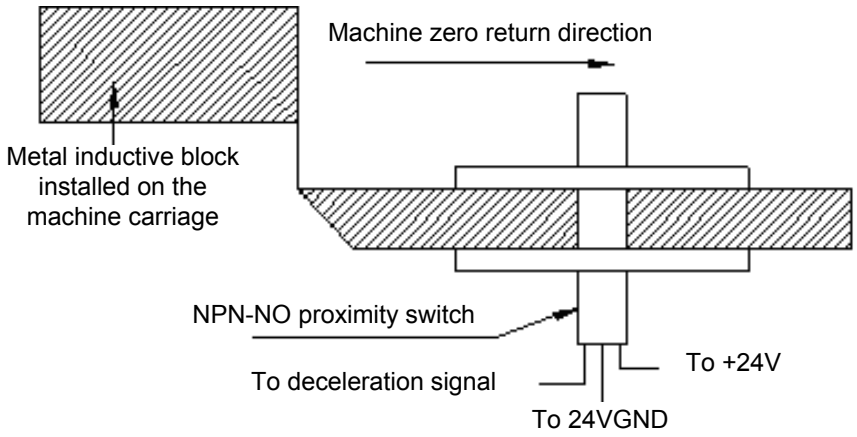


Fig. 3-2 Deceleration signal to NPN proximity switch

2) When the bit parameter **P407_d7, P407_d6, P407_d5, P407_d4, P407_d3, P407_d2** is set to 0, the machine zero return is positive, the deceleration signal LOW is valid. The following is the connection example of the deceleration signal and zero signal; the one-turn signal of servo motor is taken as the zero signal when the system is connected with the servo motor.

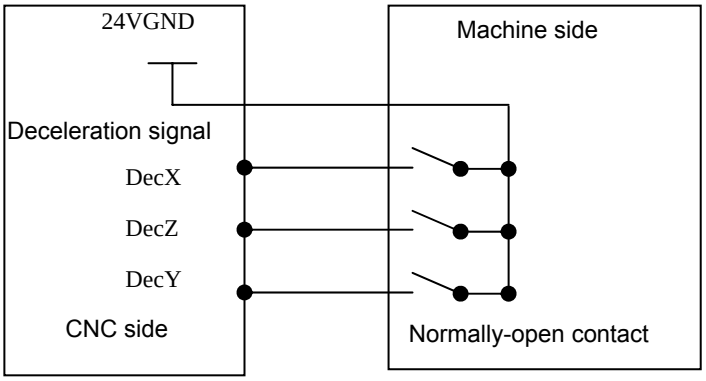


Fig. 3-3 Deceleration signal connection

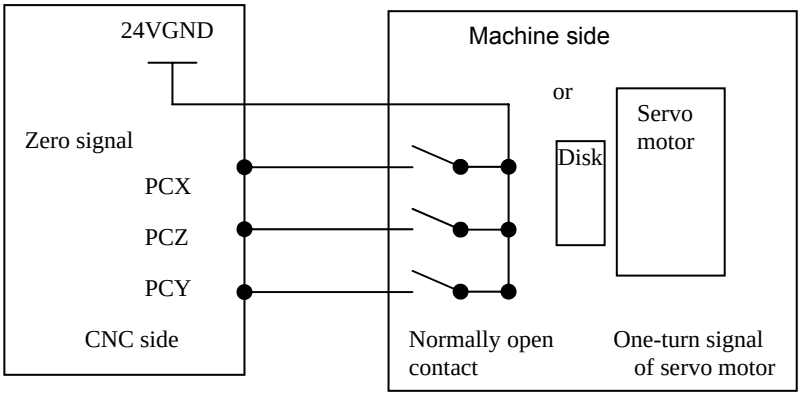


Fig. 3-4 Zero signal connection

3.4 Tool-Exchange Control Function and Connection

3.4.1 Tool-Exchange Control Signal Definition

Relative interface signals of too change control:

X2 interface	General signal name	Standard signal name	Standard signal function	Remark
Pin 8	UI08	T08	Tool post position signal T8	General input interface UI08
Pin 15	UI07	T07	Tool post position signal T7	General input interface UI07
Pin 7	UI06	T06	Tool post position signal T6	General input interface UI06
Pin 14	UI05	T05	Tool post position signal T5	General input interface UI05
Pin 12	UI04	T04	Tool post position signal T4	General input interface UI04
Pin 4	UI03	T03	Tool post position signal T3	General input interface UI03
Pin 11	UI02	T02	Tool post position signal T2	General input interface UI02
Pin 3	UI01	T01	Tool post position signal T1	General input interface UI01
Pin 1	UO12	TL+	Tool post (CW) output	As general output interface UO12
Pin 9	UO11	TL-	Tool post (CCW) output	As general output interface UO11
Pin 6	UI09	TCP	Tool post clamping in-position signal	General input interface UI09

3.4.2 Signal Connection

The connection method of common input/output (UI01～UI09,UO11～UO12) interface accords to the X1, X2 interface connection principle above. When the tool number signal is connected to the

machine external, and the tool number is in position, the corresponding tool number signal is connected to 24VGND.

3.4.3 Function Description

The tool change mode is set by P318. The tool change mode and the control time sequence are as follows:

3.4.3.1 Tool Change Mode 0

When **P318=0～8**, the tool change mode is 0, which is applicable to the line-up tool post.

3.4.3.2 Tool Change Mode 9

P318=9: tool post type 9, tool change mode 9. Use M60 to execute the tool change.

In tool change mode 9, the system automatically calls M60 to execute the tool change when T command is executed.

T command execution process as follows:

1) The system firstly modifies the tool number and the tool offset, and counts the target tool nose coordinates;

2) Modify macro variable r4005 (target tool number) and r4006(target tool offset number);

3) Call M60 to execute the tool change programs;

4) Wait M60 to be completed;

5) Complete the tool change.

【Program example】

The following is the program to compile the customized command to realize the tool change mode 9 operation; applied to the general electromotive turret tool post.

The tool signal circuit quantity separately corresponds the tool quantity defined by P319, each signal line corresponds to one tool number, and the valid level of the tool signal is LOW level. Other tool signals are invalid when a tool signal is valid.

Example: P319=4: the tool signals are as follows:

	T04	T03	T02	T01
No.1	1	1	1	0
No.2	1	1	0	1
No.3	1	0	1	1
No.4	0	1	1	1

Note: 1 in the table stands for HIGH, 0 for LOW

```
%254
M98 P1000                    ; 4tools, setting tool signal (T4 T3 T2 T1) separately in
                                 r1004~r1001
M02
```

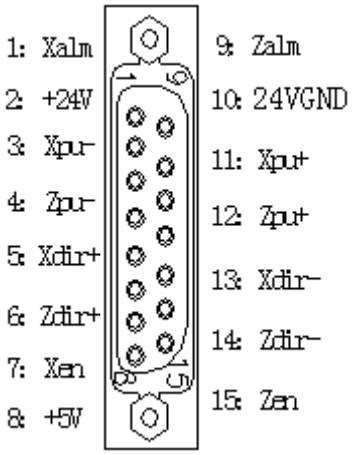
```
N1000 -M60
N1010 r1 = 14                          ; 1# tool signal (1110)
N1020 r2 = 13                          ; 2# tool signal (1101)
N1030 r3 = 11                          ; 3# tool signal (1011)
N1040 r4 = 7                           ; 4# tool signal (0111)
N1050 if(r4005 = 1) then r5 = r1       ; target tool number saved tor5
N1060 if(r4005 = 2) then r5 = r2
N1070 if(r4005 = 3) then r5 = r3
N1080 if(r4005 = 4) then r5 = r4       ; look for target tool number
N1090 r2012 = 0                ; tool post (CW) searching for target tool position
N1100 r4010 = 10000            ; set max. time (CW) to be 10m
N1110 r6 = r1000               ; read 32 input signal to r6
N1120 r7 = r6 and 15           ; read tool signal (r1004~r1001) to r7
N1130 if(r7 = r5) then P2000           ; searching for tool number
N1140 if(r4010 = 0)then P3000 ; alarm for CW overtime
N1150 M97 P1110                        ; continuously check tool signal
; tool post lock
N2000 r2012 = 1                ; stop tool post rotating after having found tool signal
N2010 r4010 = 500              ; tool post (CCW) being locked after delaying 500ms N2020
if(r4010>0) then P2020                  ; delaying wait
N2030 r4010 = 1000             ; set the CCW locking time
N2040 r2011 = 0                ; CCW locking
N2050 if(r4010>0) then P2050  ; wait the CCW locking
N2060 r2011 = 1                ; stop CCW
M99                                    ; complete the tool change
; tool change failure
N3000 r2012 = 1                ; stop tool post rotation after alarm
N3010 00S check tool selection signal overtime
N3020 r5002 = 110001                   ; set the window width to be one line
N3020 r5002 = 130168                   ; font color to be red
N3030 r5002 = 1
N3040 r5002 = 1000             ; display alarm message
N3050 M97 P3050                ; tool change failure, wait to manually stop the machining program
N3060 M99
```

In the above, No. 254 program is written to the system FLASH, and the detailed operation is referred to Program 8.2 *Customizing Commands*; and after the user sets the parameter related to the tool change mode 9, the system executes the T to realize the required tool change function in the manual tool change or the machining program, refer to the *Self-defined Command Achieve Example* in Section 8.2.3 of **PROGRAMMING**.

3.5 X3 Motor Interface

The system can be matched with the drive units of the reaction stepper motor, the compound stepper motor or AC servo motor by the motor drive unit interface X3.

3.5.1 Signal Definition



X3 motor (DB15 male)

Pin	Standard signal name	Standard signal name explanation	Signal direction
1	XALM	X drive unit alarm input terminal	Drive unit → CNC
2	+24V	X/Z drive power supply +24V	
3	XP-	X pulse positive terminal	CNC →Drive unit
4	ZP-	Z pulse positive terminal	CNC →Drive unit
5	XD+	X positive terminal	CNC →Drive unit
6	ZD+	Z positive terminal	CNC →Drive unit
7	XEN	X axis enabled	CNC →Drive unit
8	+5V	X/Z drive power supply +5V	
9	ZALM	Z drive unit alarm input terminal	Drive unit → CNC
10	0V	Power ground	
11	XP+	X pulse negative terminal	CNC →Drive unit
12	ZP+	Z pulse negative terminal	CNC →Drive unit
13	XD-	X negative terminal	CNC →Drive unit
14	ZD-	Z negative terminal	CNC →Drive unit
15	ZEN	Z axis enabled	CNC →Drive unit

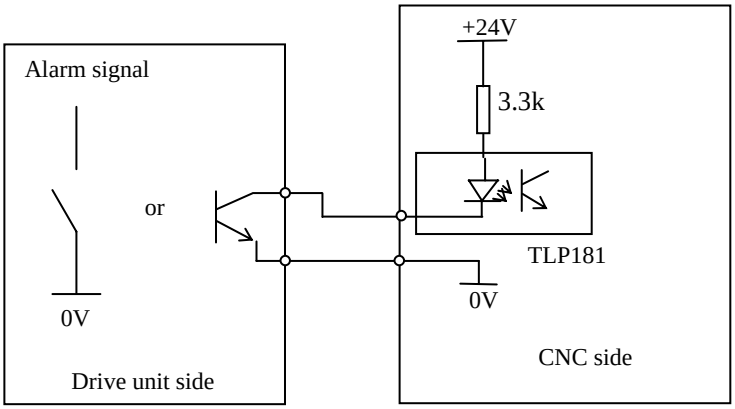
3.5.2 Technical Specifications

- Max. pulse output frequency: 500kHZ
- Pulse width:：2μs

3.5.3 Equivalent Circuit

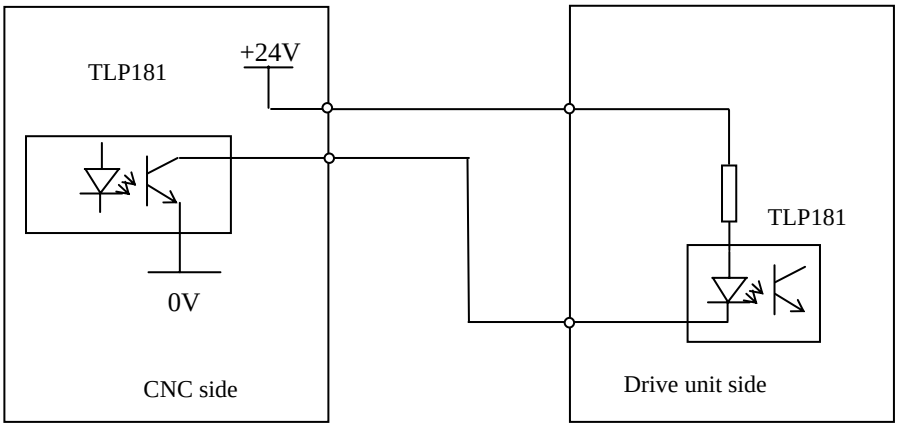
3.5.3.1 Drive Unit Alarm Signal XALM, ZALM, YALM

P405_d4,P405_d3, P405_d2 set the drive unit alarm level is LOW or HIGH. The drive unit must use the following methods to provide the signals:



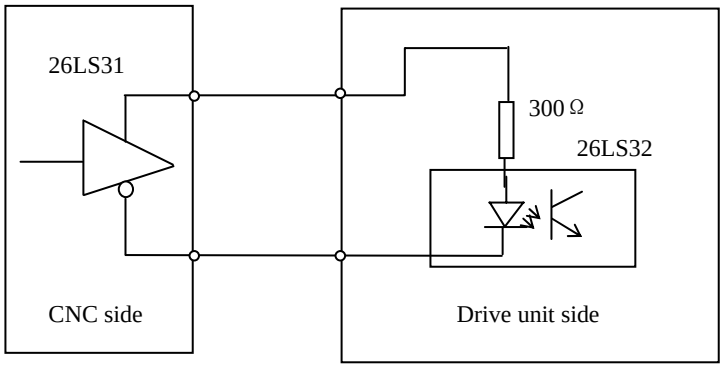
3.5.3.2 Enable Signal Xen, Zen, Yen

When the CNC is working normally, the enable signal output is valid (the enable signal is connected with 0V). The circuit diagram is shown as follows:



3.5.3.3 Pulse Signal and Direction Signal

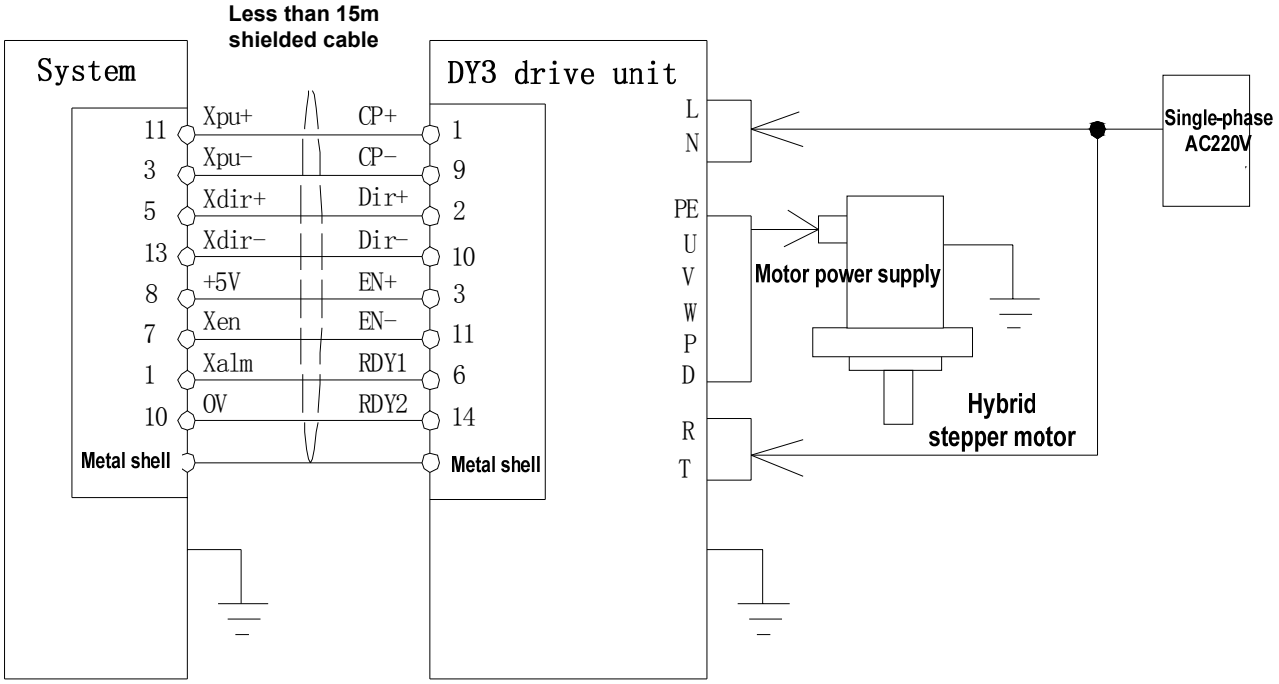
XP+,XP-,ZP+,ZP- are pulse signals, Xdir+, Xdir-, Zdir+, Zdir- are direction signals. The signals in the two groups are difference output, the external should use 26LS32 and the circuit method layout is as follows:



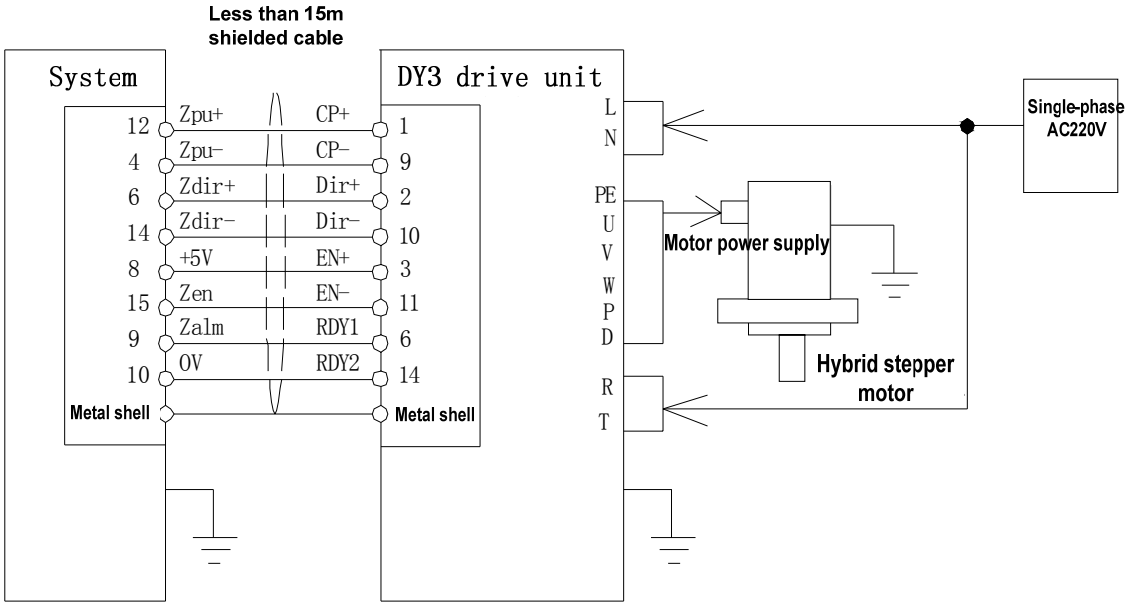
3.5.4 Connection Between CNC System and Drive Unit of Compound Stepper Motor

Connection layout between CNC and GSK DY3:

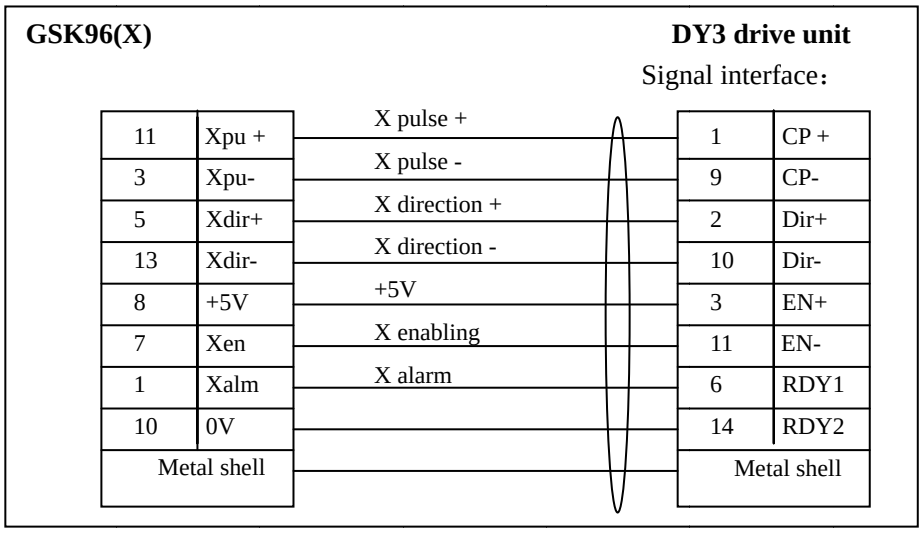
X connection layout：

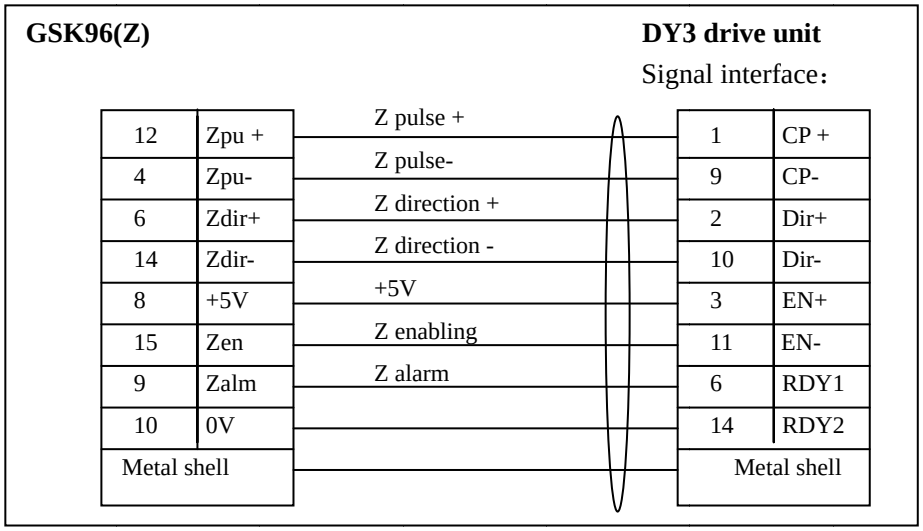


Z connection layout：



Connection table between CNC and **GSK DY3**:





When other stepper drive unit is connected with the CNC system, the system can uses the corresponding control switching, and its detailed connection method is referred to the corresponding drive device user manual.

[Notice]

When the stepper motor is used, it is regulated according to the bit parameter **P405** (**d 7～d 0**), X-motion parameter(**P100～P116**), and definitions of parameter are referred to **Operation**, **Parameter Working Mode.**

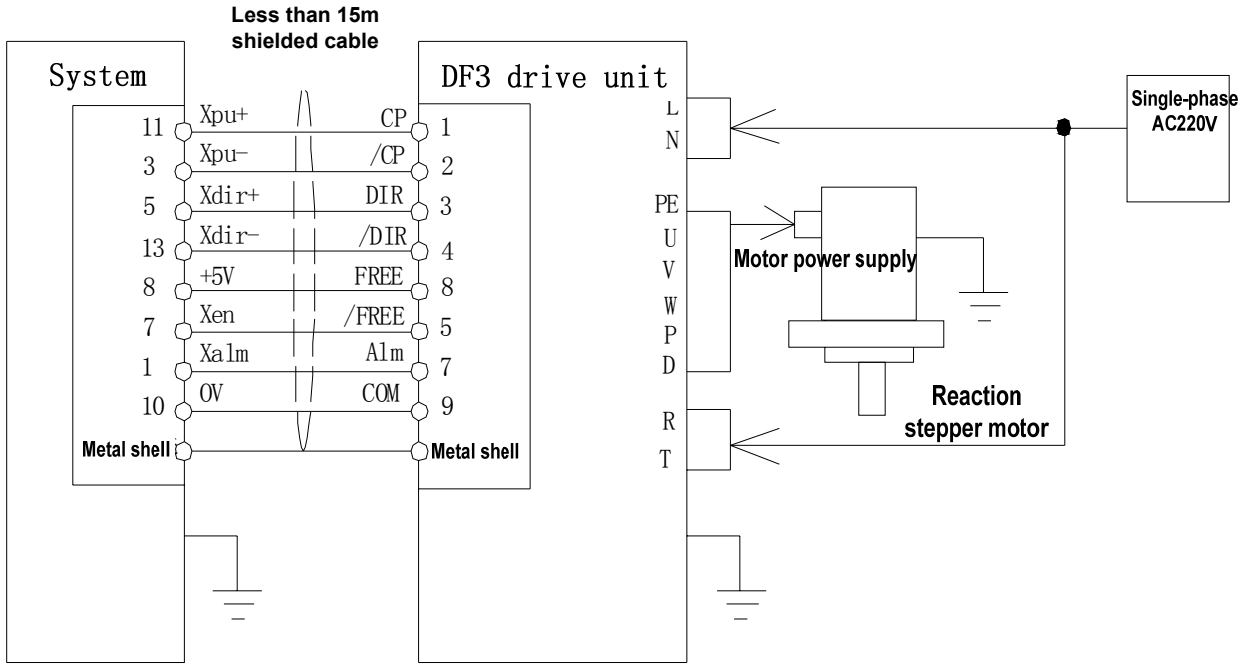
! The shield cable must be used to connect the step driver and CNC system, otherwise cause that the motor steps out owing to the external interference.

CNC system, the stepper driver and the stepper motor must be reliably connected with the earthing to avoid the motor stepping out because of the external interference.

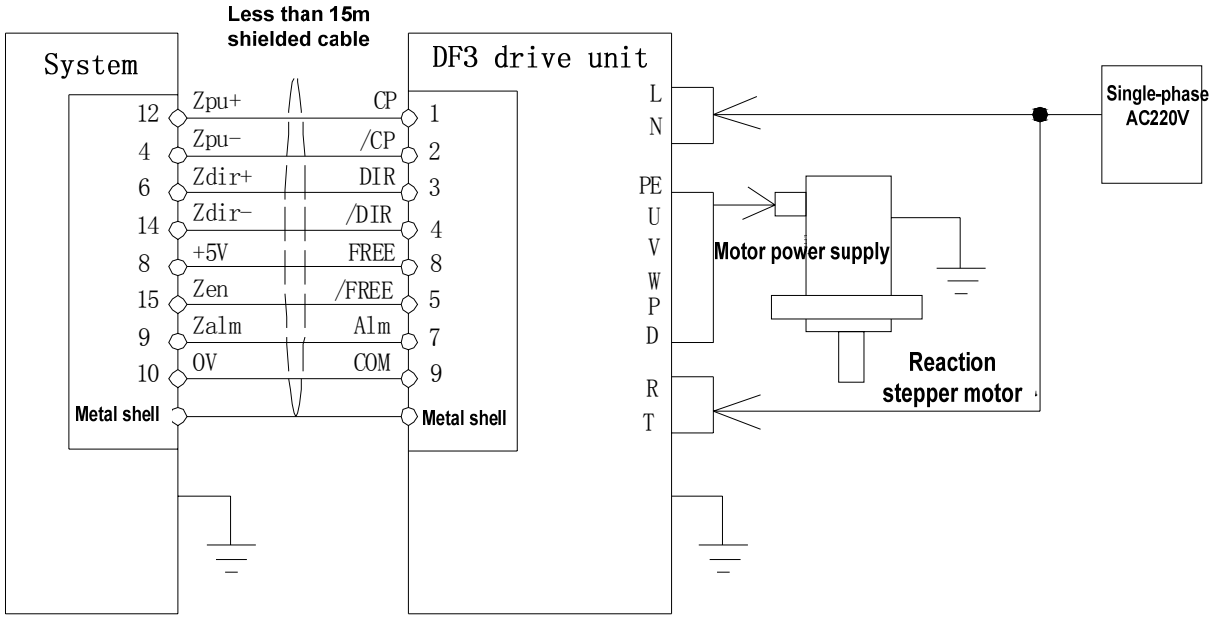
3.5.5 Connecting Between CNC and Drive Unit of Reaction Stepper Motor

Connection layout between CNC and **GSK DF3**:

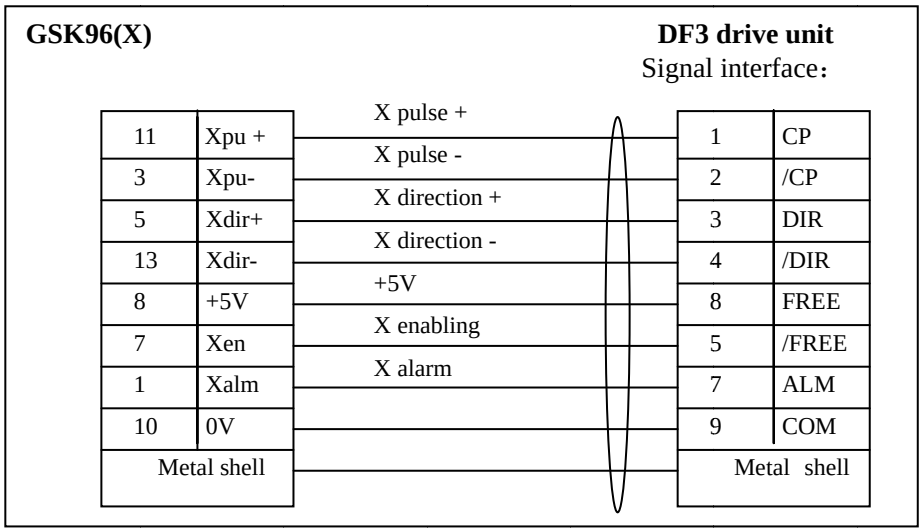
X connection layout：

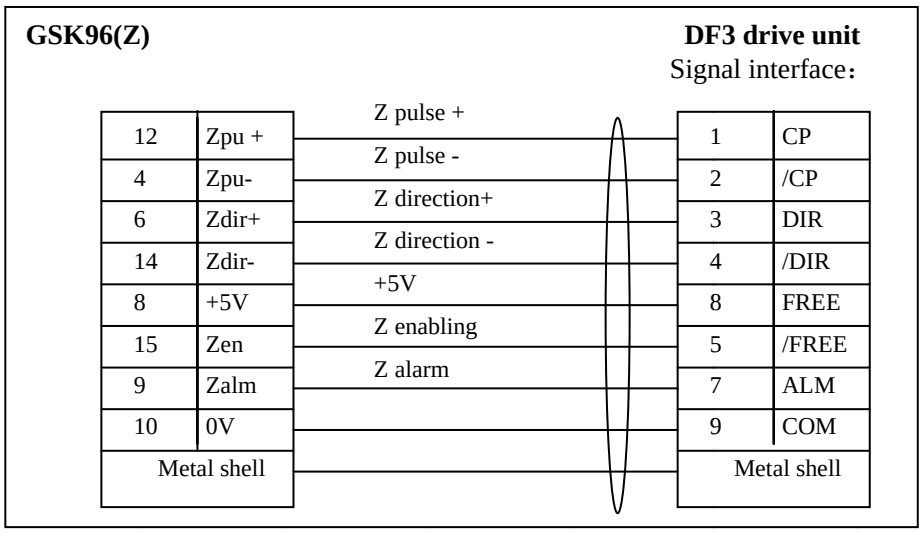


Z connection layout：



Connection table between CNC and **GSK DF3**:





[Notice]

When the stepper motor is used, it is regulated according to the bit parameter **P405** (**d7～d 0**) ,X-motion parameter(**P100～P116**) , and definitions of parameter are referred to **Operation**, **Parameter Working Mode.**

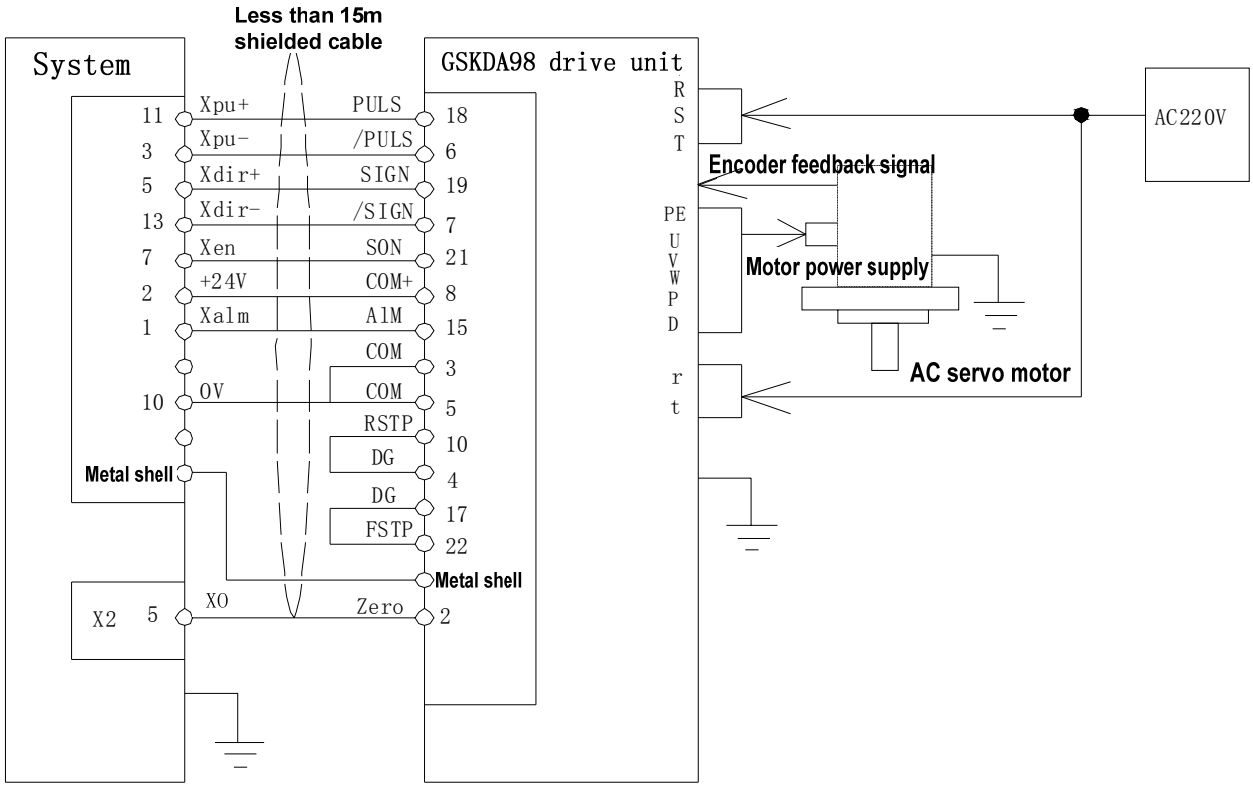
! The shield cable must be used to connect the step driver and CNC system, otherwise cause that the motor steps out owing to the external interference.

CNC system, the stepper driver and the stepper motor must be reliably connected with the earthing to avoid the motor stepping out because of the external interference.

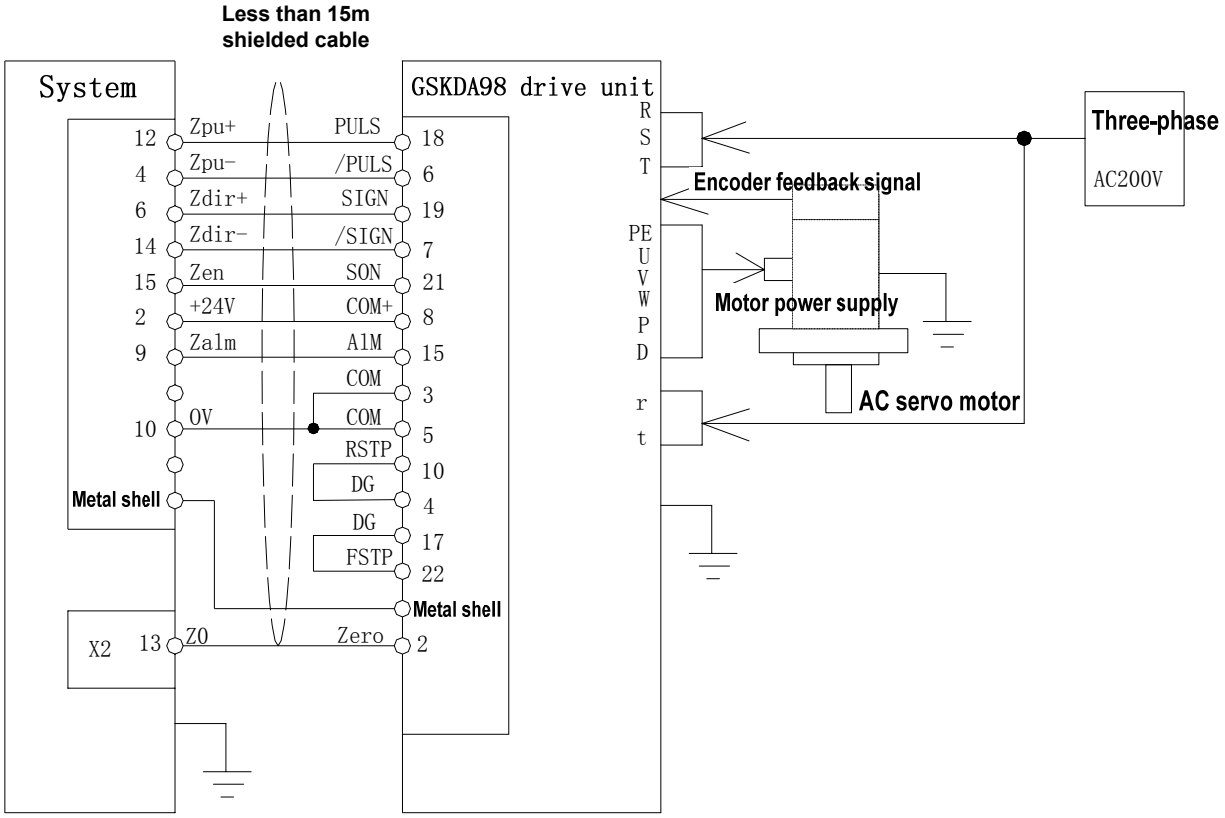
3.5.6 Connection Layout Between CNC and AC Servo Drive Unit

Connection layout between CNC and **GSK DA98 AC** servo drive unit:

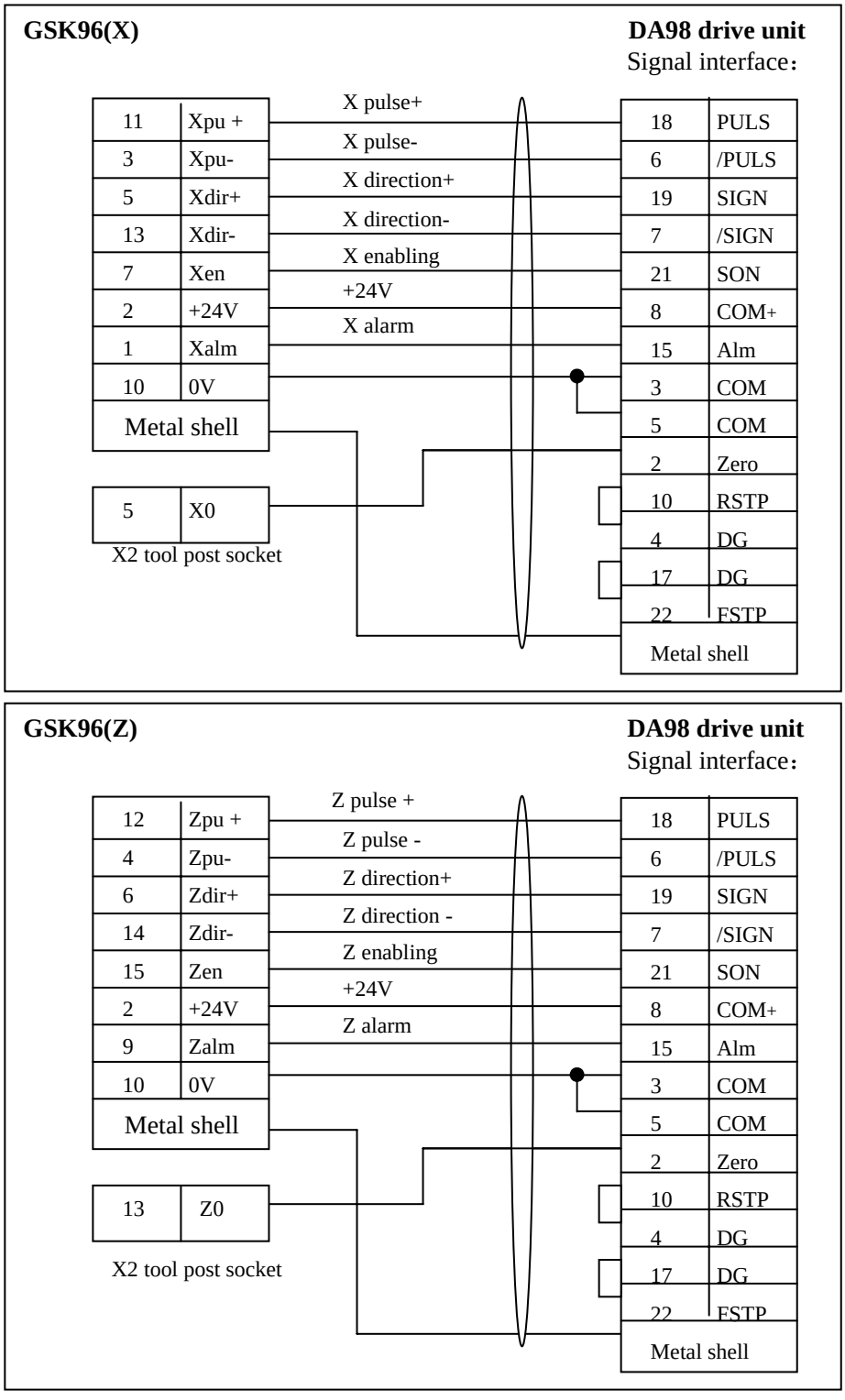
X connection layout：



Z connection layout：



Connection table between CNC and GSK DA98 AC servo drive unit.



[Notice]

When the stepper motor is used, it is regulated according to the bit parameter **P405** (**d7～d 0**) ,X-motion parameter(**P100～P116**) , and definitions of parameter are referred to **Operation**, **Parameter Working Mode.**

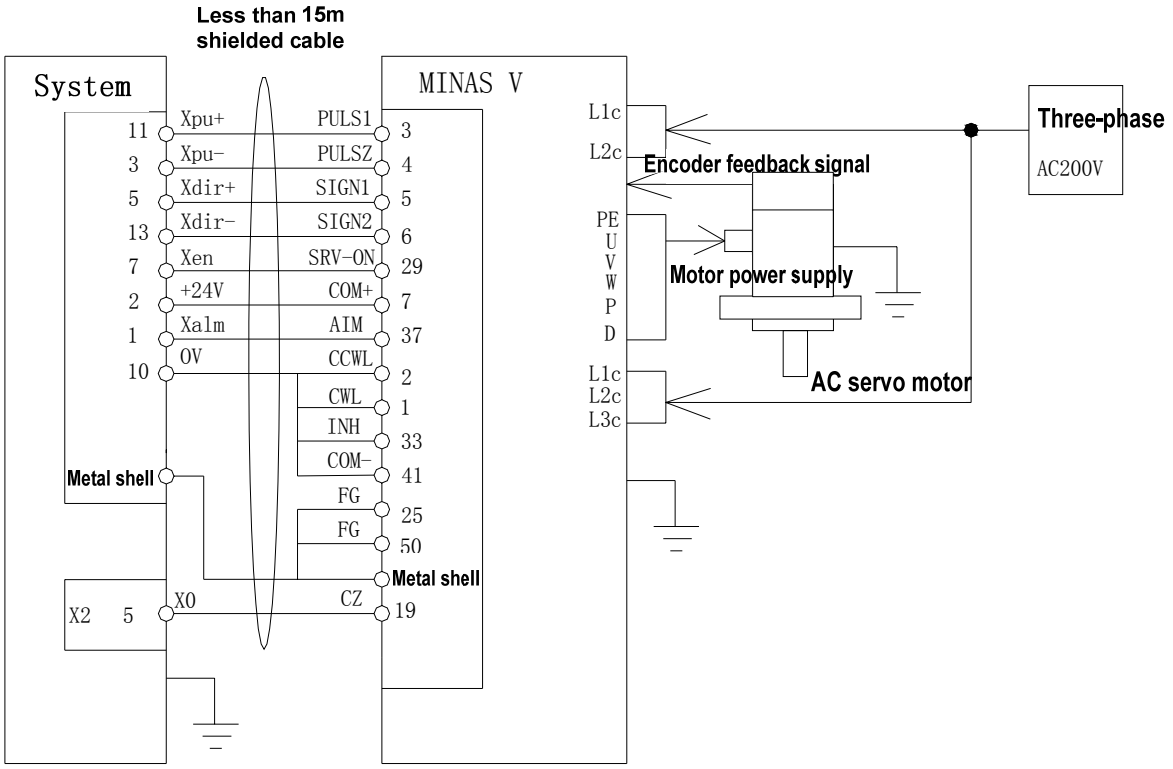
The shield cable must be used to connect the step driver and CNC system, otherwise cause that the motor steps out owing to the external interference.

CNC system, the stepper driver and the stepper motor must be reliably connected with the earthing to avoid the motor stepping out because of the external interference.

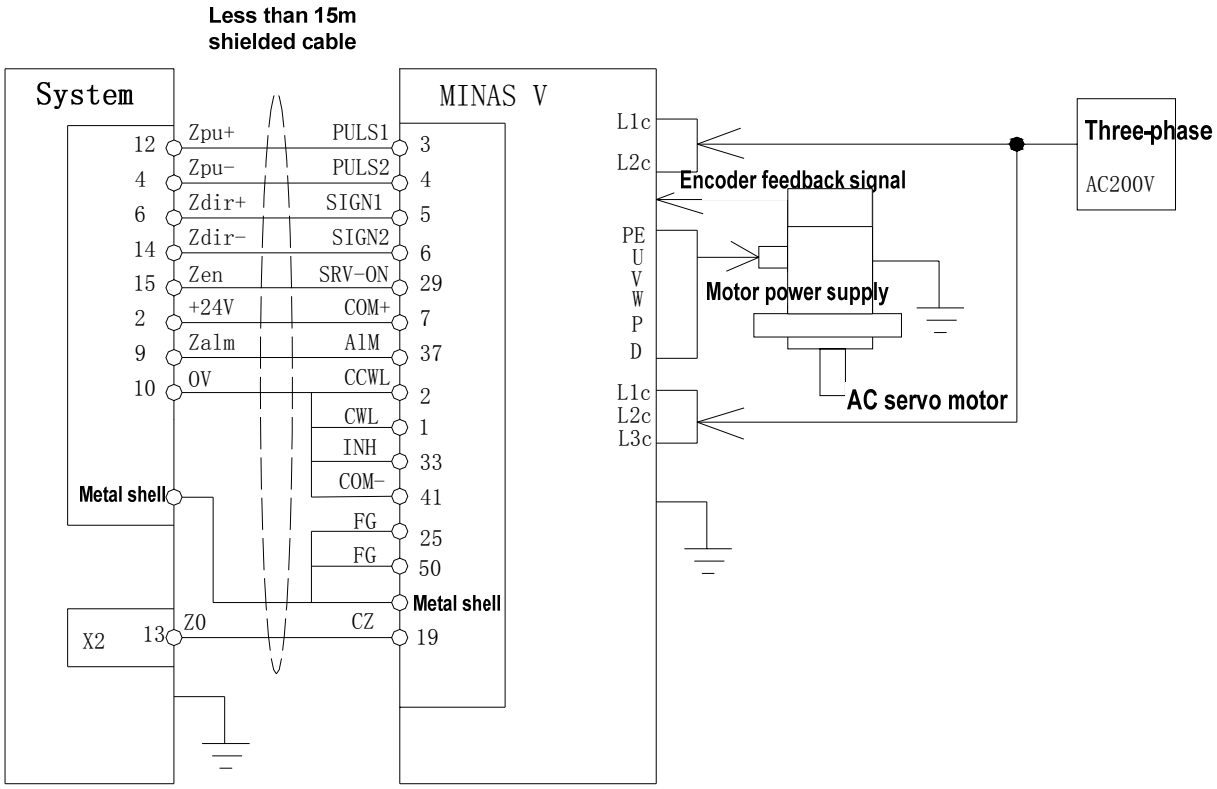
3.5.7 Connection Layout Between CNC and Panasonic Drive Unit

Connection layout between CNC and Panasonic MINAS V serial drive unit

X connection layout (Reference only):

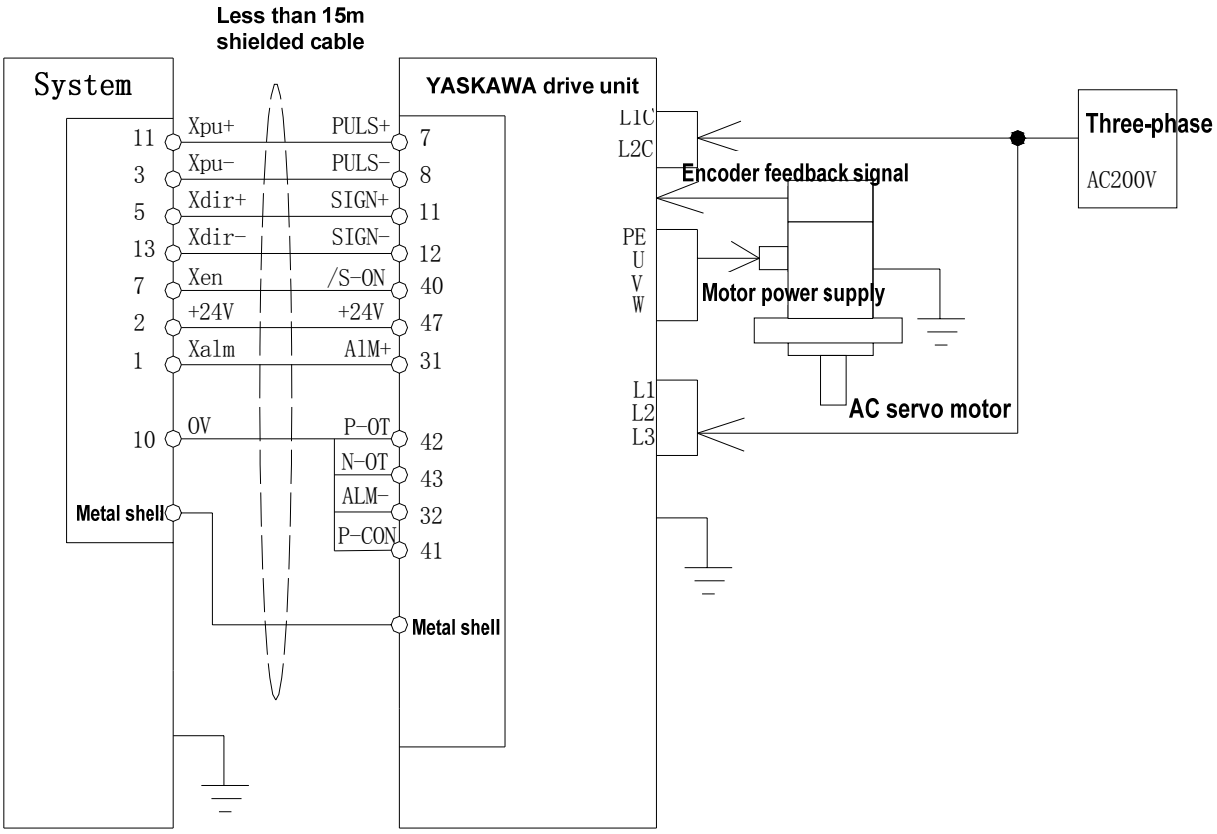


Z connection layout:

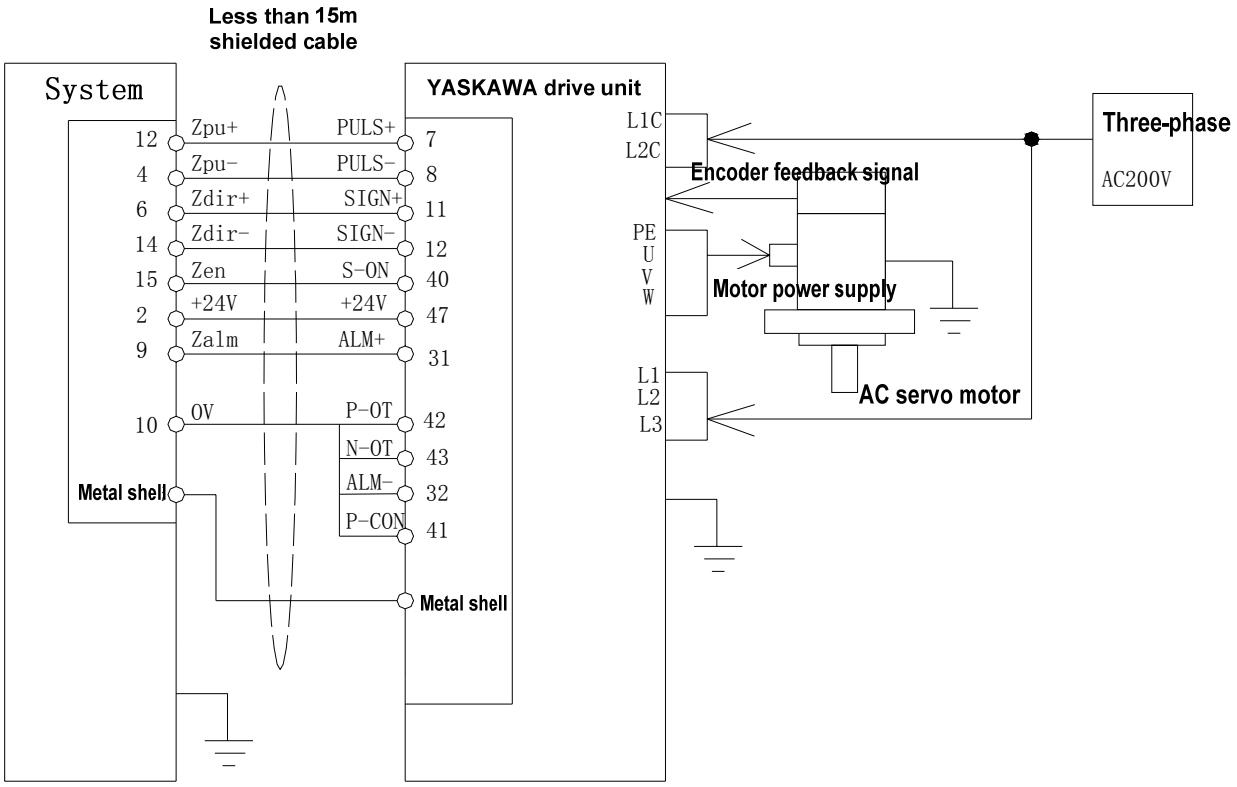


3.5.8 Connection Layout Between CNC System and Japanese Yaskawa Drive Unit

X connection layout：



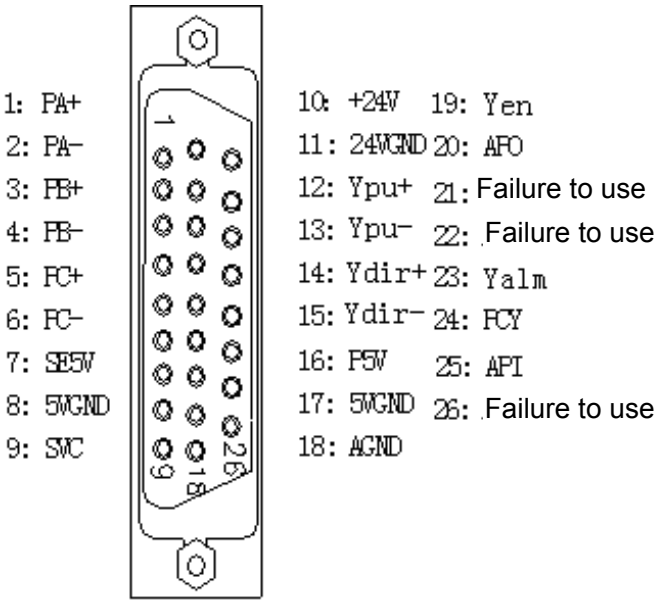
Z connection layout：



3.6 X4 Spindle Interface and Y interface

The CNC system controls the connection between the voltage (SVC) output signal and the spindle converter by the spindle X4 converter, which realizes the stepless change-speed within limits.

3.6.1 Signal Definitions



X4 spindle and Y axis (DB26 Female)

Pin name	Standard signal	Pin function
1	PA+	Encoder A pulse +
2	PA-	Encoder A pulse -
3	PB+	Encoder B pulse +
4	PB-	Encoder B pulse -
5	PC+	Encoder C pulse +
6	PC-	Encoder C pulse -
7	SE5V	Encoder 5V
8	0V (5VGND)	Encoder 5V earthing
9	SVC	Spindle analog voltage+
10	+24V	24V
11	0V (24VGND)	24V earthing
12	YP+	Y pulse +
13	YP-	Y pulse -
14	YD+	Y + direction
15	YD-	Y- direction
16	P5V	Y drive unit 5V
17	0V (5VGND)	Y drive unit 5V earthing
18	AGND	Spindle analog voltage ground
19	Yen	Y enabling
20	APO	Speed/position switch control
21	Unused	Unused
22	Unused	Unused
23	YALM	Y axis drive unit alarm
24	PCY	Y axis zero point
25	API	Speed/position state check
26	Unused	Unused

3.6.2 Converter Technical Specification

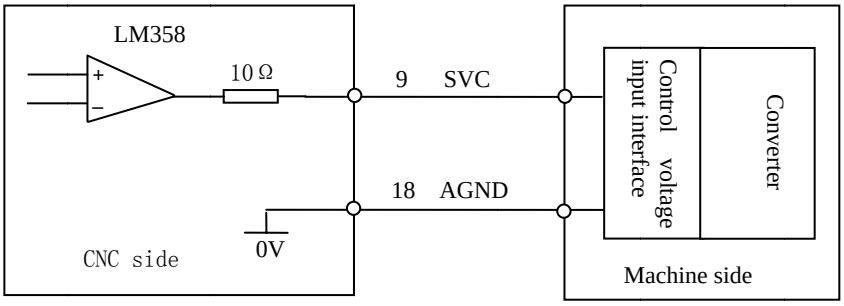
- System output analog voltage is 0 V～10 V.
- The connection cable between the system and the converter should use the shield cable, and the shield is connected with X4 socket metal shell.

3.6.3 Encoder Technical Specifications

- The system can connect externally the spindle encoder by the spindle interface X4, which is used to the thread machining, and tapping.
- It can be used the encoder with multiple linear: parameter P209 setting.
- Voltage +5V.

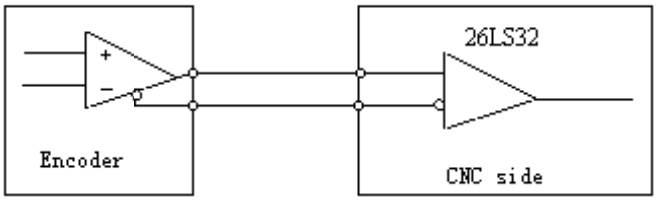
3.6.4 Connection Layout of Converter Analog Voltage

Analog spindle interface SVC can output 0～10V, and the circuit is as follow:

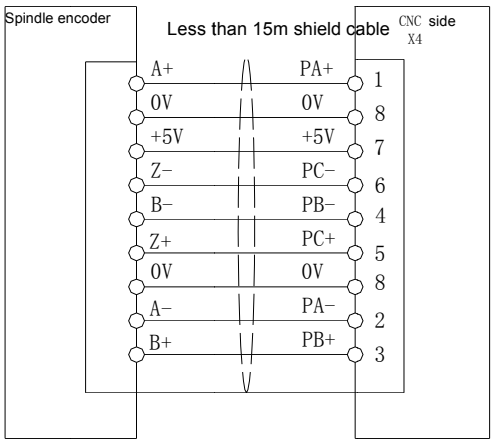


The connection cable between the system and the converter should use the shield cable.

3.6.5 Encoder Interface Method



3.6.6 Encoder Interface Connection Layout

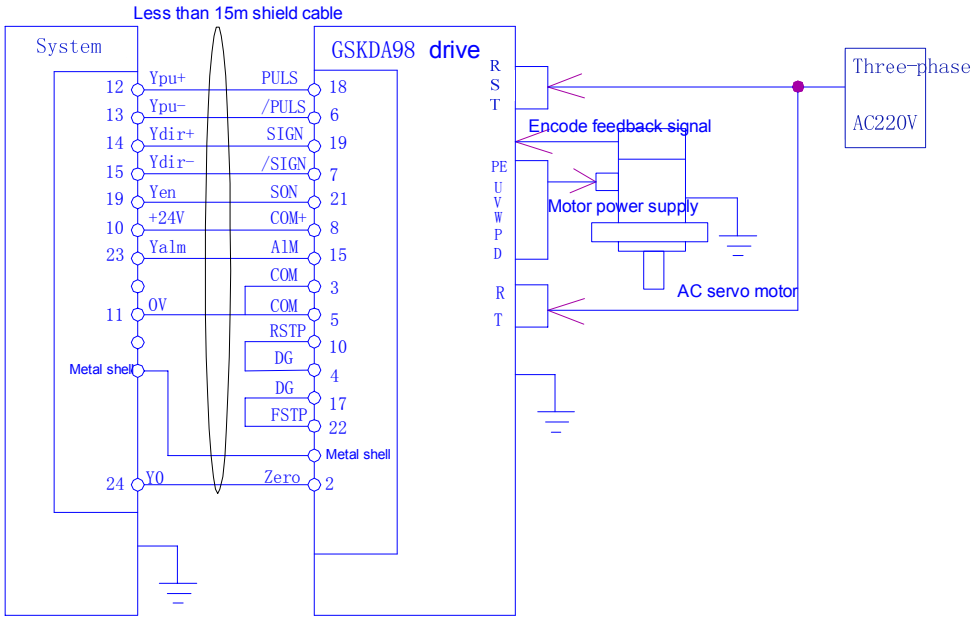


The connection cable between the system and the spindle encoder must use the shielding cable which should be connected with the socket shell.

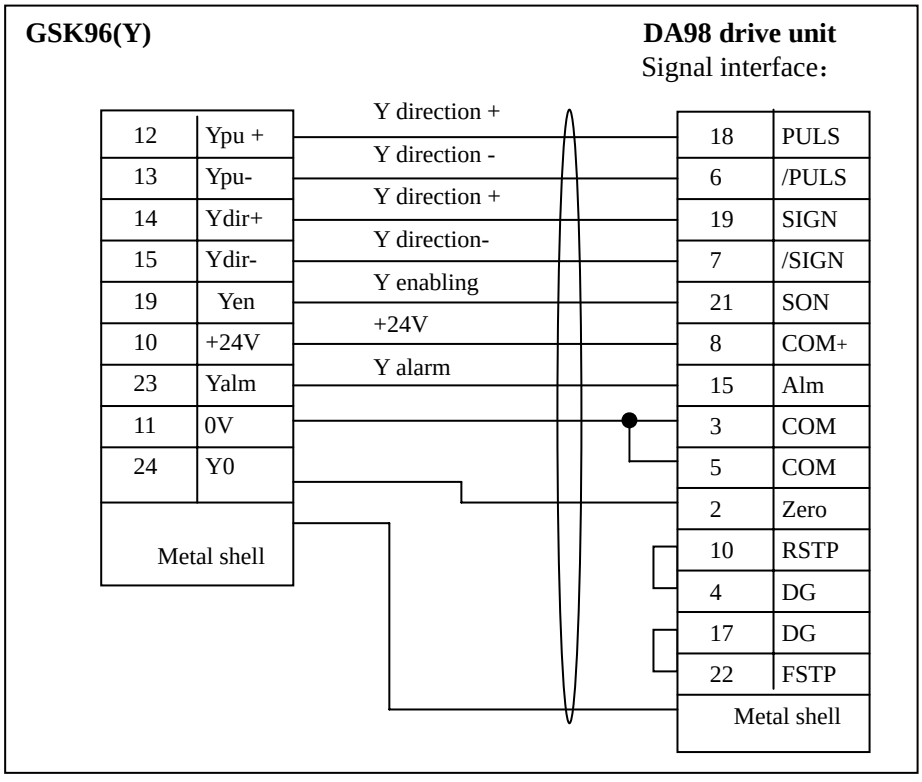
When the spindle encoder output signal is not the differential output mode, PA-, PB-, PC- cannot be connected, at the moment, the anti-interference ability of the encoder output signal reduces. It is suggested that the system should use the spindle encoder with the difference output mode.

3.6.7 Connection Between CNC System Y and AC Servo Drive Unit

Connection layout between CNC system Y and GSK DA98 AC servo drive unit



Connection table between CNC system Y and GSK DA98 AC servo drive unit



Note: When the spindle and Y are switched (P410__d4=1), Y is connected with the drive unit; the connection method between Y and other drive units is referred to that between Z/X and other drive units.

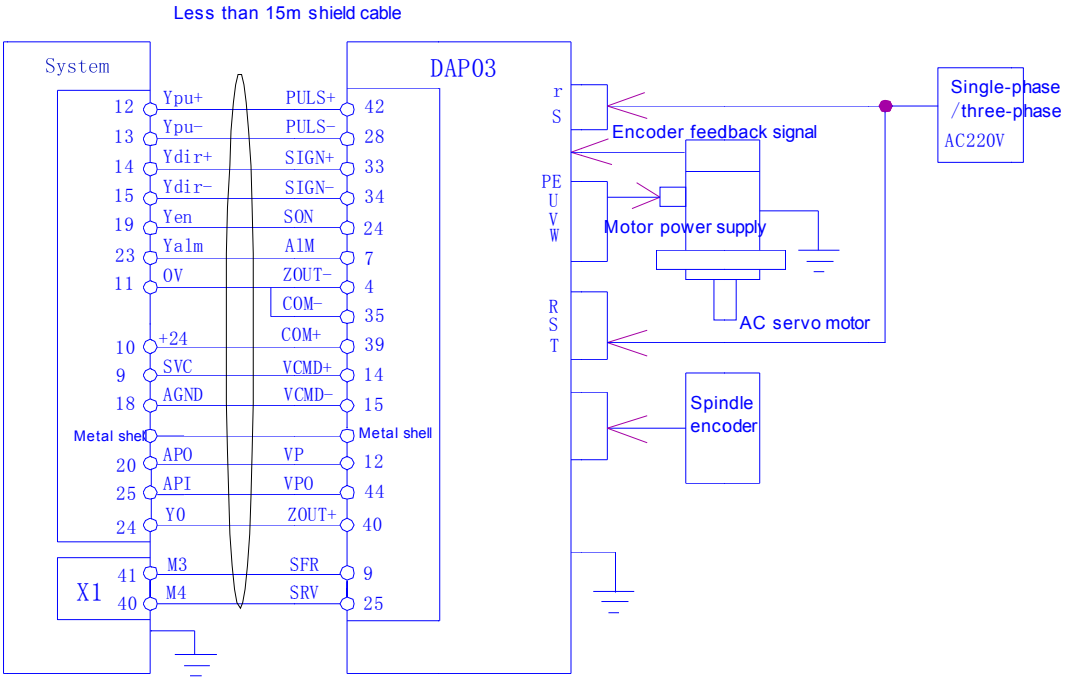
【Note】

When the stepper motor is used, it is regulated according to the bit parameter **P405**(**d7～d0**),X-motion parameter(**P100～P116**), and definitions of parameter are referred to **Operation, Parameter Working Mode.**

! The shield cable must be used to connect the step driver and CNC system, otherwise cause that the motor steps out owing to the external interference.

CNC system, the stepper driver and the stepper motor must be reliably connected with the earthing to avoid the motor stepping out because of the external interference.

3.6.8 Connection Between CNC System Y and DAP03 Spindle Drive Unit



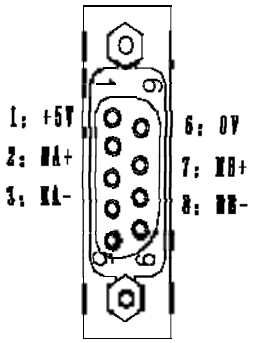
Connection table between CNC system and DAP03 spindle drive unit

GSK96(Y)			DAP03 drive unit Signal interface:	
12	Ypu +	Y pulse+	42	PULS
13	Ypu-	Y pulse-	28	/PULS
14	Ydir+	Y direction +	33	SIGN
15	Ydir-	Y direction-	34	/SIGN
19	Yen	Y enabling	24	SON
10	+24V	+24V	39	COM+
23	Yalm	Y alarm	7	Alm
			4	COM
11	0V		35	COM
9	SVC		14	VCMD+
18	AGND		15	VCMD-
20	APO		12	VP
25	API		44	VPO
X1/41	M3		9	SFR
X1/40	M4		25	SRV
24	Y0		19	ZOUT+
Metal shell			Metal shell	

3.7 X5 MPG Interface

The CNC system can be externally connected with MPG by the MPG interface X5. MPG can control the coordinate axis move. When the MPG connection line is less than 1m, it can use the single-terminal connection, when it is longer than 1.5m, it is advised to use the differential connection to improve the anti-interference ability. The input pins are not connected when the MPG has no MA-, MB-.

3.7.1 Signal Definition

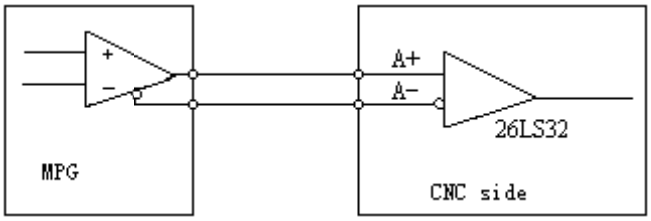


X5 MPG (DB9 Female)

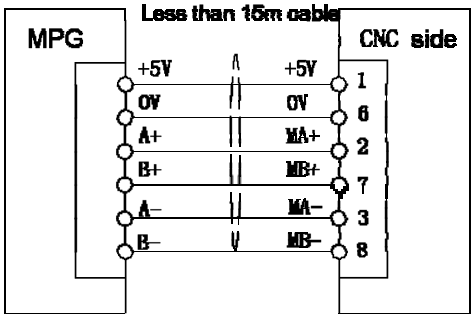
Pin	Standard signal name	Pin function
1	+5V	MPG 5V
2	MA+	MPG A pulse+
3	MA-	MPG A pulse -
4	Empty	
5	Empty	
6	0V	
7	MB+	MPG B pulse +
8	MB-	MPG B pulse -
9	Empty	

3.7.2 Interface Circuit Principle

The circuit diagram is as follows:



3.7.3 Connection Layout



When the axis moving is controlled by MPG, the moving direction of MPG cannot be changed quickly; otherwise the moving distance does not coincide with the MPG dial;

It should adopt the shield cable between the system and MPG;

When the MPG output signal is not in the difference output mode，MA- is not connected with MB-;

CHAPTER FOUR USE AND MAINTENANCE

4.1 Environmental Condition

System storage, transportation and working environmental condition as follows:

Item	Working climate condition	Storage, transportation climate condition
Environmental temperature	0℃～45℃	-40℃～70℃
Relative humidity	≤90%RH(no dewing)	≤95%(40℃)

4.2 Earthing

The correct earthing in the electricity device is important and its aims are as follows:

1) Protect the operator from being hurt because of the abnormal conditions;

2) Protect the electric devices from interference caused by the machine and electric devices nearby. (The interference may cause abnormality of the control device).

3) The machine and system must be grounded firmly, the neutral wiring in the charged barded wire net must not be the earthing line, otherwise, which causes the injury of persons or the device damaged.

4.3 Power Supply Requirements

The system can normally run in the following AC input power supply:

Voltage fluctuation: AC (0.85～1.1) ×220V

Frequency fluctuation: 50Hz ± 2%

The requirements for power supply of machine tool are described in machine tool installation manual.

4.4 Guard

The guard grade is not less than IP20.

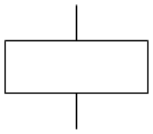
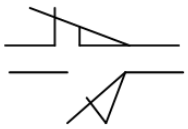
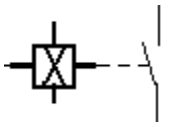
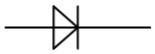
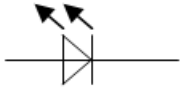
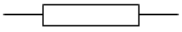
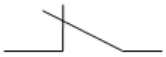
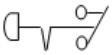
4.5 Reuse it Again

After the CNC system is not used for long time, the user must firstly clean the dust, and dry up it, and then check the wiring, the earthing of the CNC system device, keeps it power-on in some time to ensure that the system runs without any failure.

APPENDIX

Appendix 1 CNC system electrical component symbol explanations

The following examples are some connection symbols of electrical components.

Name	Symbol	Graph	Name	Symbol	Graph
Relay	**K**		Travel switch	**S**	
Motor	**M**		Hall proximity switch	**B**	
Semiconductor diode	**D**		Indicator	**E**	
Capacitor	**C**		LED	**P**	
Resistor	**R**		Normally-open contact		
Push-button switch with lock	**S**		Normally-closed contact		
Push-button switch without lock	**S**				

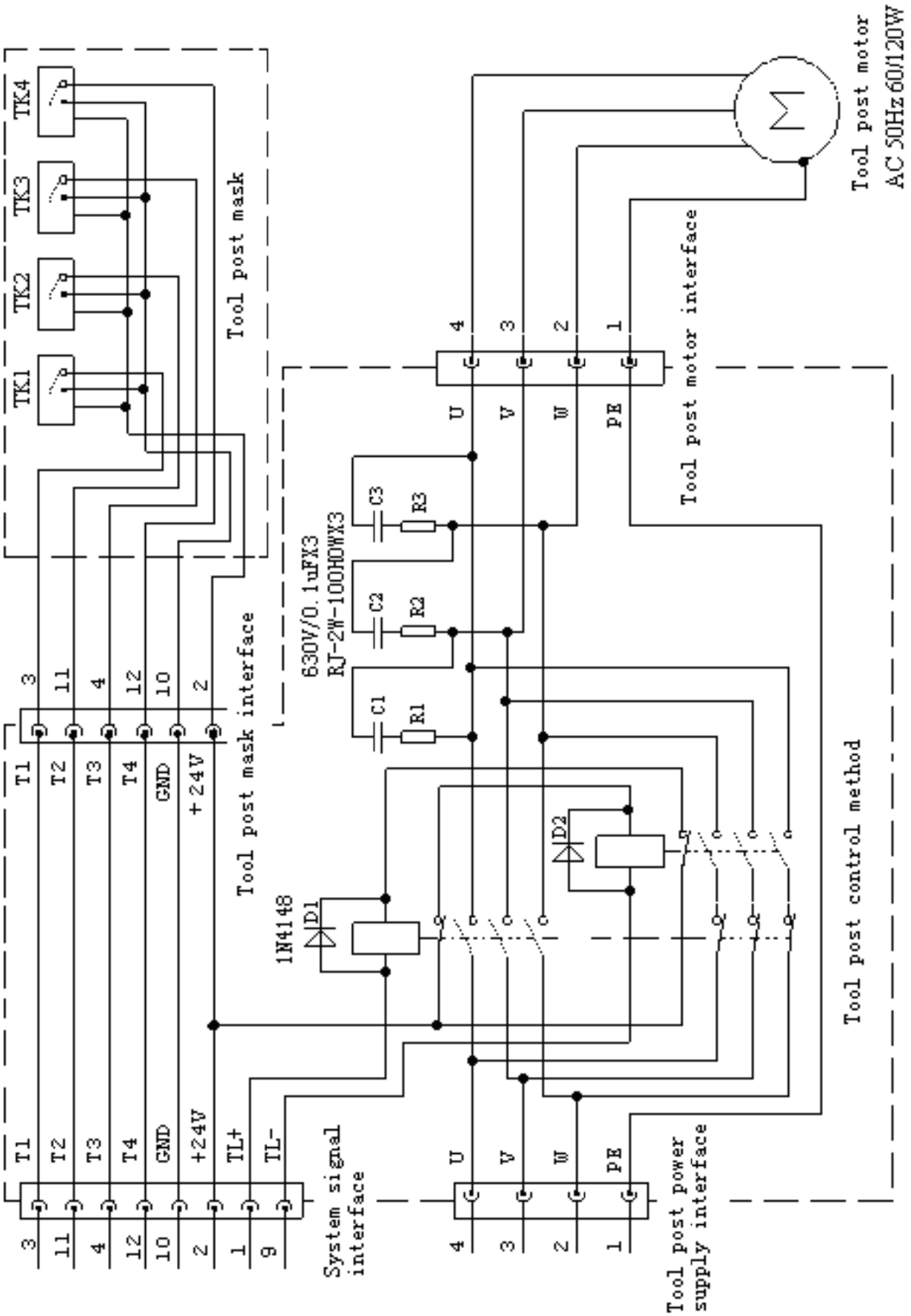
Shield layer

Protective earthing

Exchange.

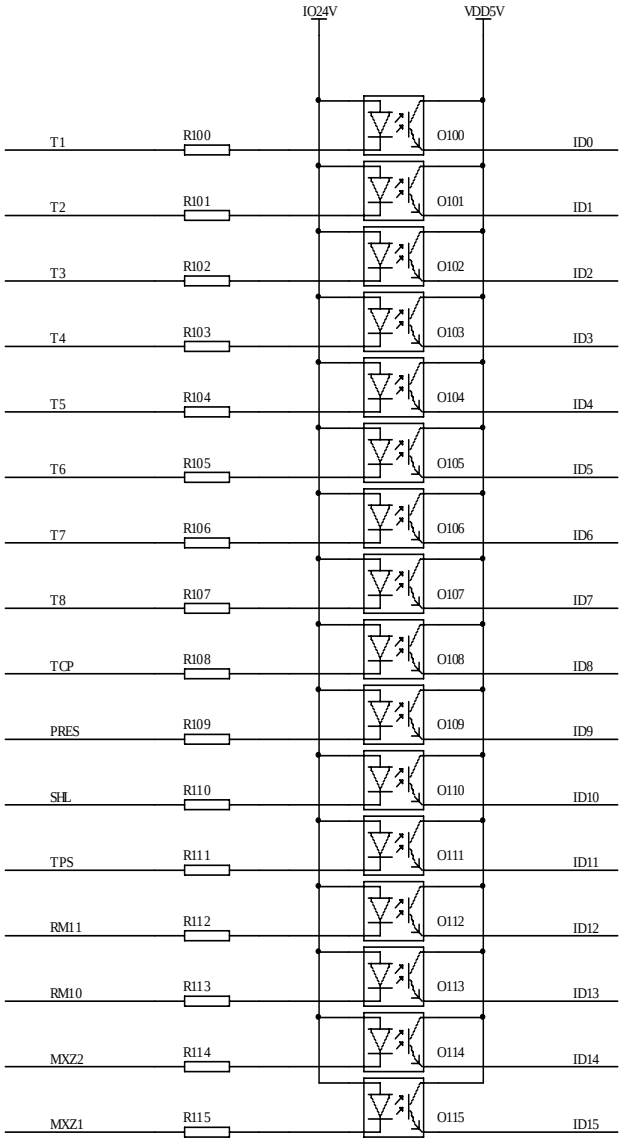
Wiring terminal

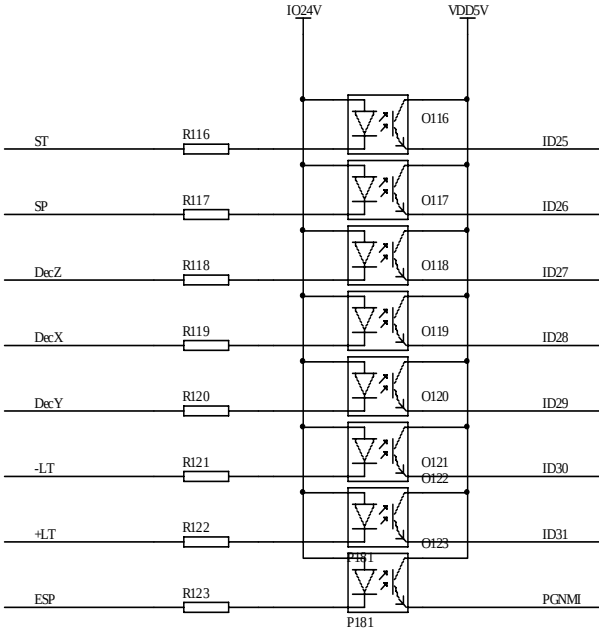
Appendix 2 CNC system tool post controller circuit method layout

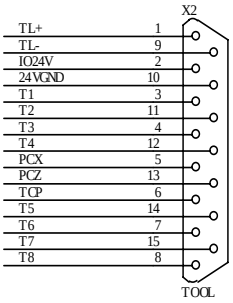


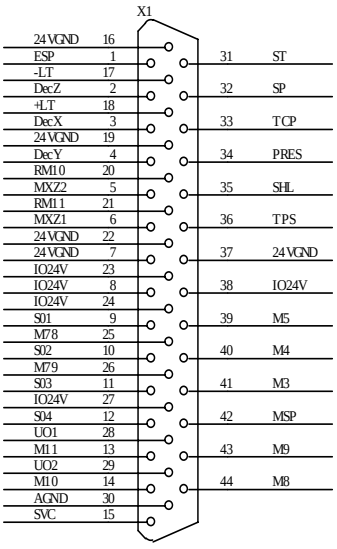
Appendix 3 Interface circuit method layout

Interface circuit method layout 1 (input)

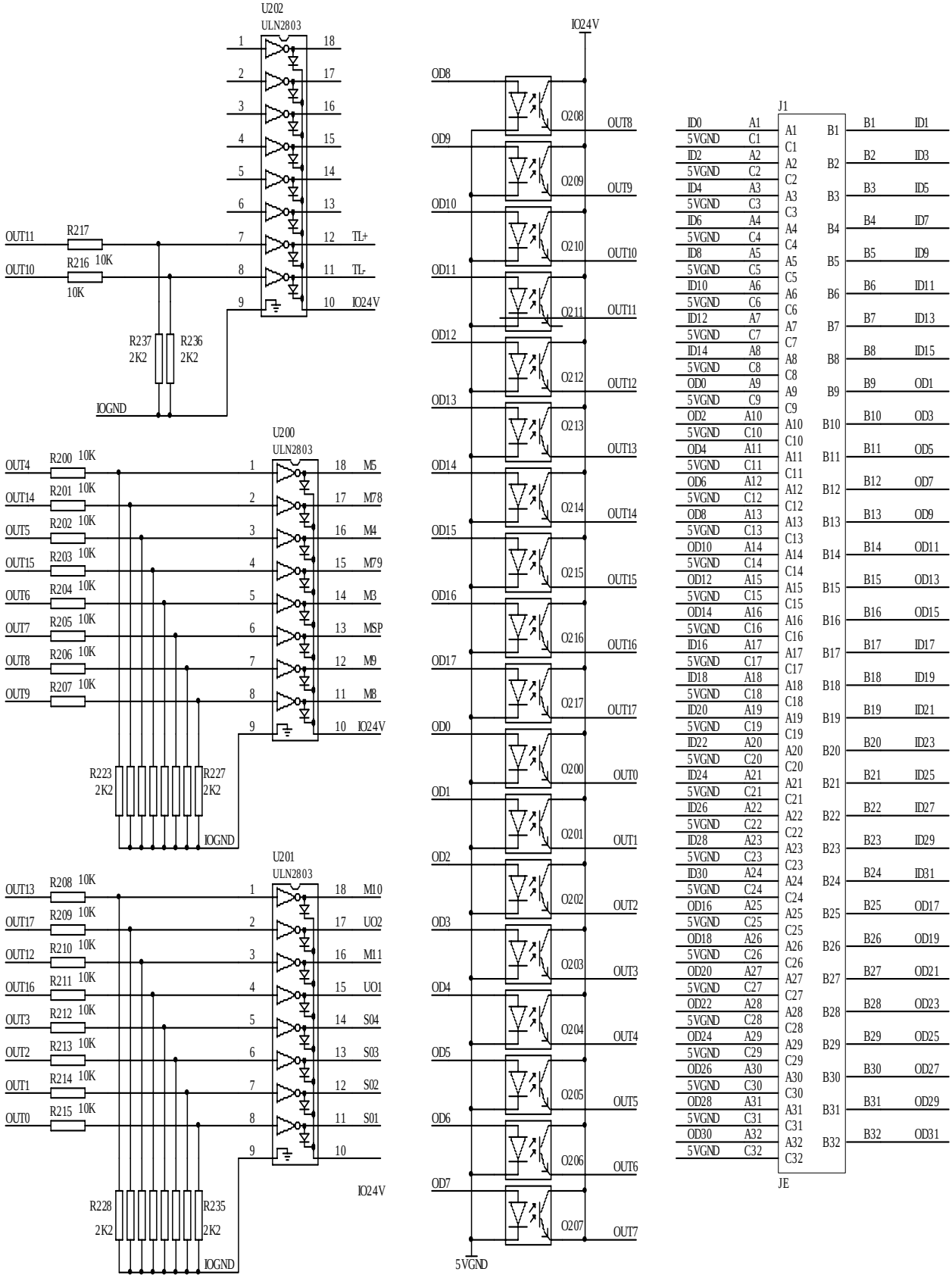




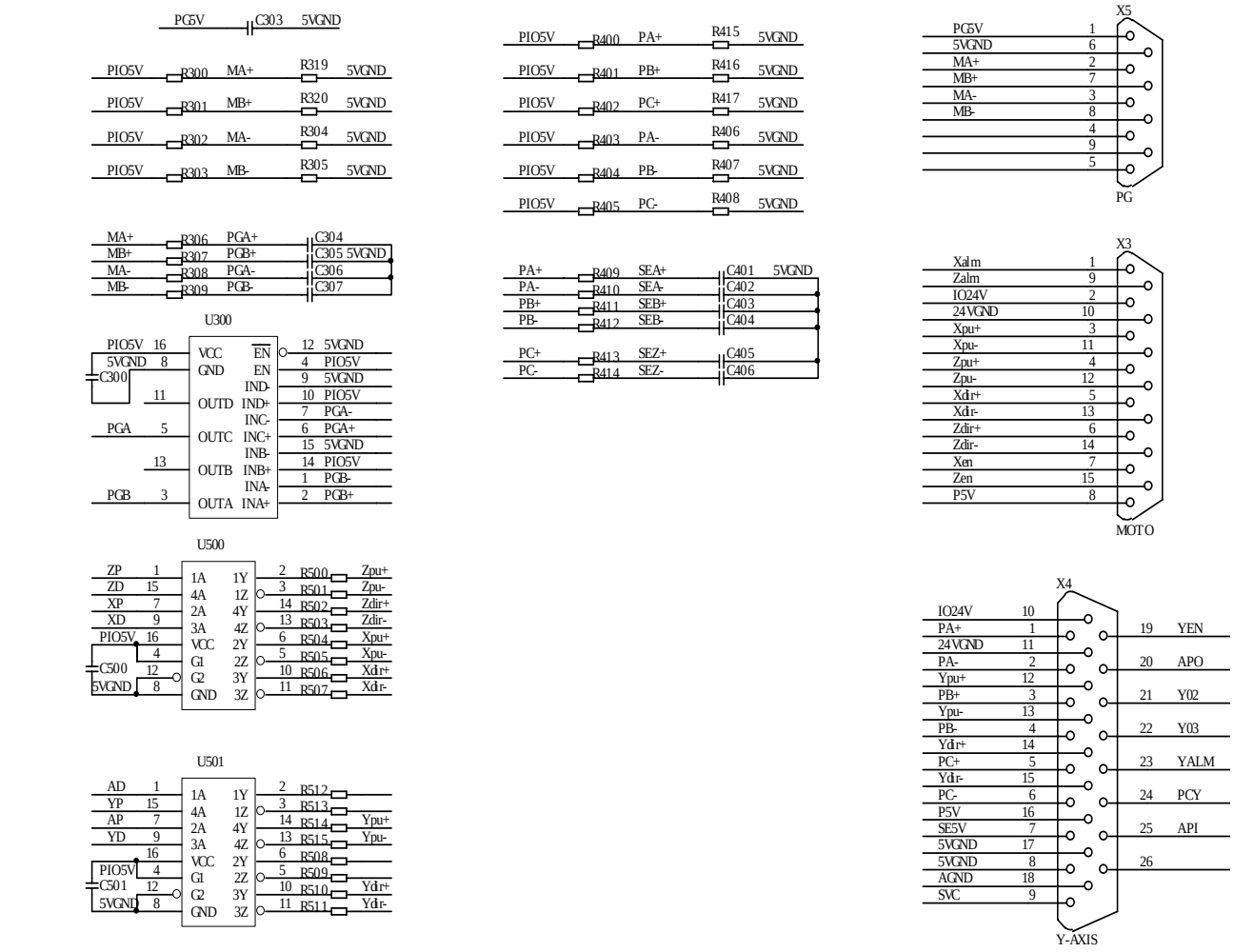


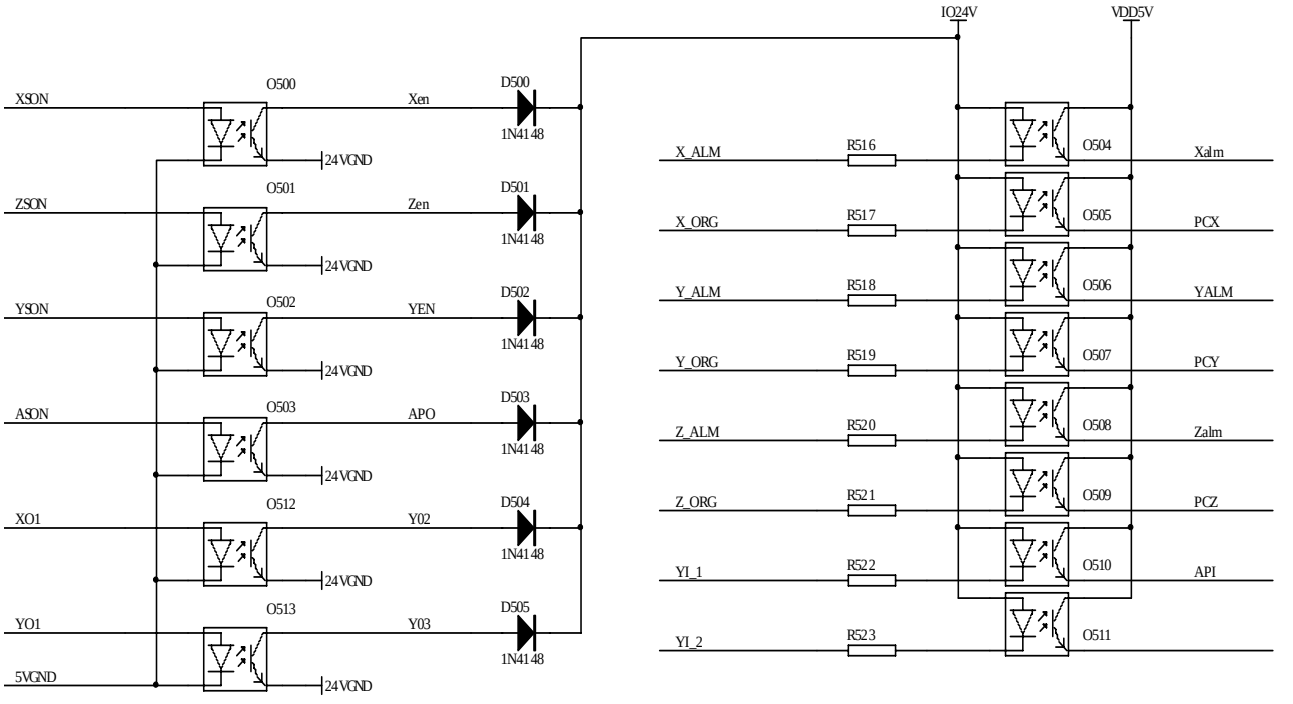


Interface circuit method layout 2 (output)

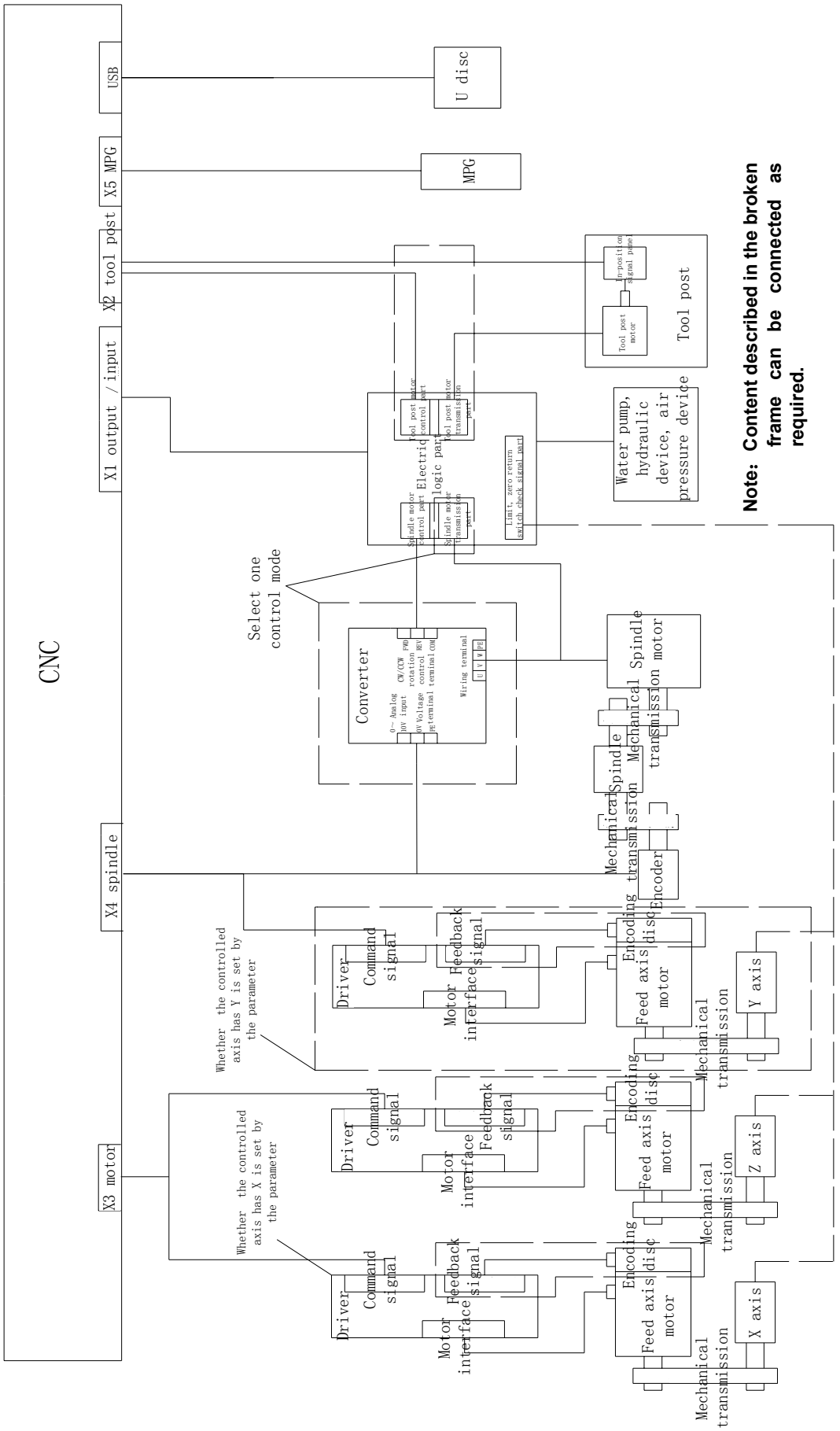


Interface circuit method layout 3





Appendix 4 External control connection layout



Note: Content described in the broken frame can be connected as required.

CNC external control connection

Appendix 5 CNC system appearance installation dimension